SmartPlant Instrumentation

Tutorial



Version 2009 SP2 (9.0.2)

October 2009/December 2010

DSPID2-PB-200010A-Updated

INTERGRAPH®

Copyright

Copyright © 1995-2009 Intergraph Corporation. All Rights Reserved.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.
Intergraph Corporation
P.O. Box 240000
Huntsville, AL 35813
Street address: 170 Graphics Drive, Madison, AL 35758

Terms of Use

Use of this software product is subject to the End User License Agreement ("EULA") delivered with this software product unless the licensee has a valid signed license for this software product with Intergraph Corporation. If the licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives licensee permission to print a reasonable number of copies of the documentation as defined in the applicable license agreement and delivered with the software product for licensee's internal, non-commercial use. The documentation may not be printed for resale or redistribution.

Warranties and Liabilities

All warranties given by Intergraph Corporation about equipment or software are set forth in the EULA provided with the software or applicable license for the software product signed by Intergraph Corporation, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license. No responsibility is assumed by Intergraph for the use or reliability of software on equipment that is not supplied by Intergraph or its affiliated companies. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Trademarks

Intergraph, the Intergraph logo, PDS, SmartPlant, SmartMarine, FrameWorks, I-Convert, I-Export, I-Sketch, IntelliShip, INtools, ISOGEN, MARIAN, SmartSketch, SPOOLGEN, SupportManager, and SupportModeler are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Microsoft and Windows are registered trademarks of Microsoft Corporation. MicroStation is a registered trademark of Bentley Systems, Inc. Other brands and product names are trademarks of their respective owners.

Contents

Introduction **11**

Administration Tasks 11
System Administrator Activities 11
Domain Administrator Activities 12
General Tasks 12
Instrument Engineering Tasks 12
Creating Instruments and Control Loops 12
Defining Process Data, Performing Calculations, Creating Specifications 12
Performing Wiring Operations 13
Generating Loop Drawings 13
Generating Hook-Up Drawings 13
Demo P&ID 14
Wiring Block Diagram 15

Administration Tasks **17**

System Administrator Activities 17
Task 1 — Initializing a Domain in the Database 18
Task 2 — Defining SmartPlant Instrumentation Users and the Domain Administrator 20
Domain Administrator Activities 22
Task 1 — Log on as Domain Administrator 23
Task 2 — Defining Domain Administrator Access Rights 23
Task 3 — Creating a Plant Hierarchy 25
Task 4 — Defining Instrument Tag and Loop Naming Conventions 27
Task 5 — Creating Custom Fields and Tables 29
Task 6 — Organizing Users into Groups and Granting Access Rights 30
Task 7 — Setting Report Management Options 36
Task 8 — Performing Miscellaneous Activities 37

Getting Started with SmartPlant Instrumentation **39**

Task 1 — Logging on to SmartPlant Instrumentation 39
Task 2 — Defining Default SmartPlant Instrumentation Properties 40
Define Units of Measure and Accuracy 40
Set SmartPlant Instrumentation Preferences 41

Creating Instruments and Control Loops **43**

Task 1 — Defining Instrument Type Profiles 43
Define Instrument Types 43
Define Reference Wiring for a Field Device 44
Define the FT (D/P Type Flow Transmitter) Instrument Type Profile 48
Define the FY and TY (I/P Transducer) Instrument Type Profiles 51
Define the FV and LV (CONTROL VALVE) Instrument Type Profiles 54
Define Additional Instrument Type Profiles 55
Task 2 — Using Supporting Tables to Add Values to Select Lists 56
Create a P&ID Drawing Document Number 56
Create Lines (Piping) 57

Enter Instrument Status Data 59
Enter I/O Type Data 59
Enter Location Data 60
Enter Manufacturer Data 60
Enter Model Data 60
Enter Values in the Custom Tables 61
Task 3 — Creating Loops with Tags Using the Domain Explorer 61
Define a Loop and Add Tag Numbers 62
Add a Tag Number to an Existing Loop 67
Assign Custom Table and Custom Field Values to a Tag Number 68
Task 4 — Creating a Loop with Its Tag Using the Instrument Index Module 69
Create a Loop with a Single Instrument Tag 69
Task 5 — Duplicating a Loop with Its Tag Numbers 72
Duplicate a Loop 73
Task 6 — Creating Loops and Tag Numbers in Batch Mode 75
Create Typical Loop FL-1 76
Create Typical Loop T-10 77
Create Plant Loops in Batch Mode 78
Task 7 — Creating Additional Loops and Viewing Instrument Index Data 79
Create Additional Plant Loops and Instruments 79
Search for a Specific Loop and View Loop Tag Properties 80
View Plant Loops, Instruments, and Associated Items 81

Viewing and Modifying Data for Multiple Records 83

Task 1 — Viewing and Editing Instrument Properties from a Browser View 83
View and Edit Data for a Single Tag Number in the Instrument Index Standard Browser 83
Edit the Browser View Window Records – Multi-Row Editing 84
Task 2 — Customizing a Browser View 84
Customize the Browser View 84
Task 3 — Copying and Pasting Values in the Browser View Window 88
Copy a Specific Value to Several Fields 88
Copy Values to Multiple Fields 88
Task 4 — Sorting and Filtering Data for the Browser View 89
Create a Customized Sorting Sequence 90
Create a Customized Filter 90
Open the Sorted and Filtered Browser 91

Defining Process Data 93

Task 1 — Defining Process Data for Lines 93
Create a Line 93
Define Line Process Data 93
Copy Process Data to Another Line 95
Task 2 — Defining Process Data for Instruments 96
Define Process Data for a Flowmeter 96
Define Process Data for a Control Valve 98

Performing Calculations and Sizing 101

Task 1 — Calculating Flowmeter Parameters 101
Calculate Flowmeter Parameters 101
Task 2 — Calculating and Sizing Control Valve Parameters 103

Calculate and Size Control Valve Parameters 103

Working with Specifications 105

Task 1 — Generating, Viewing, and Editing Specifications 105
Generate a Specification for a Flowmeter 105
Open a Specification for a Control Valve 107
Task 2 — Create a Specification Form Data Template 108
Create a Form Data Template for a Flow Transmitter 108
Protect Spec Data Fields from Being Overwritten 110
Copy Data from a Form Data Template to a Specification 111
Task 3 — Creating and Using a Multi-Tag (See-List) Spec 111
Create a Format for a Multi-Tag Spec 111
Generate a Multi-Tag Spec 113
Add an Existing Tag to a Multi-Tag Spec 115
Create a New Tag and Associate it with a Multi-Tag Spec 115
Task 4 — Customizing Spec Pages 117
Create a Customized Spec Page 117
Create a Form Containing the Modified Flow Element Page 120
Create a Spec for Flow Orifices Based on the Custom Form 121
Task 5 — Specification Revisions 122
Add Revisions to Specification Sheets 122
Generate a Comparison Report Between the Current Spec and an Archived Revision 123
Make Global Revisions for Specifications 124

Managing Documents 129

Task 1 — Creating a Specification Binder Package 129
Create a Specification Binder Package 129
Assign Specification Sheets to the Binder Package 131
Task 2 — Creating Form Notes and General Notes 132
Create a Form Note Template 132
Create a Form Note from the Template 134
Task 3 — Editing and Revising Specification Sheets from the Binder Package 135
Edit Specification Sheets 135
Revise a Binder Package for the First Time 135
Make Subsequent Changes and Revisions for a Binder Package 136
Task 4 — Printing Documents for a Specific Revision 138
Print Documents for a Particular Revision 138
Task 5 — Creating a General Document Binder Package 139
Create a General Document Binder Package 140
Add Calculation Reports to the Document Binder Package 140
Assign External Documents to the Document Binder Package 143

Performing Wiring Operations 145

Understanding Concepts and Presentation of Panel and Cable Wiring 145
Task 1 — Creating Reference Panels 147
Create Reference Junction Boxes and Terminal Strips 147
Edit Terminals 150
Create a Reference Marshaling Rack 151
Task 2 — Creating Terminal Strips within a Marshaling Rack 152
Create JB SIDE-1 153

Create Strip JB SIDE-2 by Duplicating JB SIDE-1 154
Create DCS Side Terminal Strip DCS SIDE - AI 155
Create DCS Side Terminal Strip DCS SIDE-AO by Duplicating JB SIDE-1 157
Task 3 — Creating a Reference DCS Panel 158
Create a Reference DCS Panel 158
Create a Rack in the New Reference DCS Panel 159
Create Two Additional Racks in the New Reference DCS Panel 161
Create an Analog Input I/O Card within the New Reference DCS Panel 162
Create a Terminal Strip with I/O Channels 164
Create an Analog Output I/O Card and a 4-Channel Terminal Strip within the New Reference DCS Panel 166
Task 4 — Copying the Reference Panels to the Domain Explorer 168
Create a Plant Junction Box 169
Create a Plant Marshaling Rack 171
Rename the Terminal Strips in the Domain Explorer 171
Copy Your DCS from the Reference Explorer to the Domain Explorer 171
Task 5 — Creating Reference Cables 171
Create a Reference Single-Pair Cable 172
Create a 16-Pair Cable 176
Create a Reference 8-Pair Cable and a 4-Pair Cable for Your DCS Analog Input Card Connection 177
Task 6 — Copying Reference Cables to the Plant 179
Copy Reference Cables with Sets and Wires 179
Task 7 — Making Connections 182
Connect Multi-Pair Cables to Junction Boxes 182
Filter the Cables in the Domain Explorer 184
Task 8 — Connecting Device Cables 187
Connect a Single Field Device to a Junction Box 187
Connect Field Instrument Cables in Batch Mode 188
Connect Field Side Wiring to a Marshaling Rack 191
Task 9 — DCS Management 192
Assign I/Os to the DCS 192
Effect I/O Assignment for the Analog Tag 101-FY-100 196
Connect the Analog Input Cable to the DCS Side 197
Connect the Analog Output Cable to the DCS Side 198
Connect the Analog Input and Output Cables to the Marshaling Rack Side 199
Task 10 — Cross Wiring the Signals in the Marshaling Rack 201
Cross Wire One Wire Manually 202
Cross Wire the Rest of the Signals Automatically 206
Task 11 — Adding a New Instrument to the Existing Wiring 208
Add a New Instrument to the Existing Wiring 208
Task 12 — Viewing and Printing Wiring Reports 213
View and Print Various Wiring Reports 213
Task 13 — Creating Wiring Items for a New Tag 213
Create a New Instrument Tag without an Associated Profile 213
Add a Device Panel and a Cable to the Tag 216
Task 14 — Wiring Design that Includes a Barrier 219
Define a New Apparatus Configuration 220
Create a New Apparatus and Make All the Connections 222
Task 15 — Wiring Design for Other Loop Types 226
Design the Wiring for Loop T-202 226
Design the Wiring for Loop F-201 237

Generating Loop Drawings....251

Generating Loop Drawings Using the Enhanced Report Utility....251
Task 1 — Setting Preferences....251
Set Preferences for Enhanced SmartLoop Reports....251
Apply Generation Method to Individual Loops....252
Task 2 — Generating an Enhanced SmartLoop Drawing....253
Generate an Enhanced SmartLoop Drawing....253
Change the Loop Generation Method to Show Individual Signals....256
Task 3 — Modifying an Enhanced SmartLoop Drawing....257
Modify Properties of Items from the Enhanced Report Utility....257
Open Documents Associated with Instruments....259
Modify the Wiring Connection from the Enhanced Report Utility....259
Task 4 — Changing the Enhanced Report Layout Properties....262
Define an Enhanced Report Layout and Assign the Layout to a Loop....262
Assign an Enhanced Report Layout to Multiple Loops....265
Task 5 — Adding Macros or Macro Labels to an Enhanced SmartLoop Drawing....268
Add a Macro or a Macro Label to a Tag Number....268
Attach a Macro or a Macro Label to Wiring Equipment....272
Define and Use Filters with a Macro....274
Task 6 — Adding Annotations to an Enhanced SmartLoop Report....277
Include a Watermark....277
Add Redlining....279
Attach Redlining Items to a Layout....282
Add a Macro Caption....283
Generating CAD Loop Drawings....285
Task 1 — Setting Preferences....285
Task 2 — Defining Instrument Blocks....287
Task 3 — Assigning Blocks to Instrument Types....289
Task 4 — Defining Border and Logo Blocks....291
Task 5 — Editing the Title Block Data for a Specific Loop Drawing....292
Task 6 — Macros in CAD Loop Drawings....295
Task 7 — Viewing Assigned Blocks and Using different Block Assignment Methods....297
Task 8 — Generating CAD Loop Drawings....302

Working with Hook-Ups....305

Task 1 — Setting Preferences for Enhanced Hook-Up Drawings....305
Set Preferences for Enhanced Hook-Up Drawings....305
Task 2 — Creating Hook-Up Libraries and Items....307
Create Hook-Up Item Libraries and Set One Library as Active....308
Create Hook-Up Items for Associating with the Control Valve Hook-Up....308
Create Hook-Up Items for Associating with the Flow Instrument Hook-Up....310
Generate a Report of Hook-Up Items....310
Task 3 — Creating Hook-Ups and Assigning them to Instruments....312
Create Hook-Up Types and Hook-Ups....312
Associate Hook-Up Types with Instrument Types....315
Associate Instrument Tags with Hook-Ups....316
View Drawing Symbol Content....317
Task 4 — Associating Items with the Hook-Ups....318
Assign Items from the Active Item Library to the Control Valve Hook-Up....318
Assign Items from the Active Item Library to the Flow Instrument Hook-Up....320
Task 5 — Generating a Bill of Material (BOM) and a Hook-Up Drawing....321

Generate a Bill of Material 321
Generate an Enhanced Hook-Up Drawing 322

SECTION 1

Introduction

The aim of this tutorial is to familiarize you with the basic features of SmartPlant Instrumentation® (powered by INtools®) – getting the program up and running, creating a minimal setup for your plant, and performing all the basic instrument engineering activities for each module. This tutorial will provide you with the fundamental understanding, skills, and practical experience you need to begin using SmartPlant Instrumentation with confidence.

In the course of the tutorial, you will be guided step-by-step to create a loop with its associated wiring, line, and instrument process data. You will also perform calculations and generate specification sheets, loop drawings, and installation details (hook-ups).

IMPORTANT

- We recommend that you make a backup copy of the database before using it with the tutorial.
- You must use the In_demo.db database, or if working on an Oracle or SQL Server platform, you must use In_demo.db as the source when initializing your database for the tutorial.
- You should work through all the objectives in a given task in one session.

The tutorial is divided into two parts: one for the System and Domain Administrators and the other for Instrument Engineers. If you are a novice, we strongly recommend going through the entire tutorial thoroughly.

In this tutorial, the following diagrams are included to help you build your database using typical engineering data:

- Demo P&ID 101-PID01-001
- Interconnection wiring block diagram

Throughout this tutorial, you will open new screens and use new options. While some of these are explained here in detail, we encourage you to browse through the *SmartPlant Instrumentation User's Guide* and *SmartPlant Instrumentation Online Help* to gain knowledge and help.

NOTE From Version 2007, SmartPlant Instrumentation documentation is now included with the program setup, and is installed in the same folder as the SmartPlant Instrumentation program. You can access the printable versions (PDF files) of the various installation and users' guides using the **Printable Guides** command on the SmartPlant Instrumentation **Help** menu.

Administration Tasks

System Administrator Activities

The System Administration tasks that you will learn include:

- How to initialize a domain.
- How to add users to the SmartPlant Instrumentation user list (requires creation of a department).
- How to create and assign a Domain Administrator for the domain.

Domain Administrator Activities

The Domain Administration tasks that you will learn include:

- How to set up the plant hierarchy.
- How to create hierarchy items and define tag and loop naming conventions.
- How to define a group and assign users to the group.
- How to define access rights for a group.
- How to add custom fields and tables to the Instrument Index.

General Tasks

The general tasks that you will perform include:

- How to open a specific unit in SmartPlant Instrumentation.
- How to define default units of measure and preferences.

Instrument Engineering Tasks

Creating Instruments and Control Loops

The tasks in this section include:

- How to define data in supporting tables, including instrument type profiles, P&ID drawing numbers, and lines.
- How to create loops and associated tags.
- How to view and edit instrument data using the Instrument Index Standard Browser.
- How to generate and print out reports.

Defining Process Data, Performing Calculations, Creating Specifications

In these sections, you will learn how to define process data, how to use those values to perform calculations that generate additional data, and how to generate specifications that include process data and calculation results. The tasks in these sections include:

- How to create lines.
- How to define line process data.
- How to define instrument process data for flow elements, control valves, and so forth.
- How to perform instrument sizing calculations.
- How to generate single-tag and multi-tag (see-list) specifications that include process data and calculation data.
- How to copy data to specifications
- How to modify specification pages
- How to perform revisions on specifications.

Performing Wiring Operations

In this section, you will create the essential wiring items and make the connections needed to specify a complete control loop for the tag numbers that you created. You will also learn how the software can create a signal from the field instruments to the control panel.

The tasks in this section include:

- How to create panel-strip-terminal reference items and copy them to your plant.
- How to create cable-set-wire reference items and copy them to your plant.
- How to assign tags to DCS channels.
- How to connect cables / wires as appropriate.
- How to design the wiring for specific control loops, including connection type definitions and cross-wiring connections.

Generating Loop Drawings

In this section, you will learn how to generate and manage enhanced SmartLoop drawings using the Enhanced Report Utility. Also, you learn how to assign CAD drawing files to instruments and then generate loop drawings in a CAD application (SmartSketch).

The tasks in this section include:

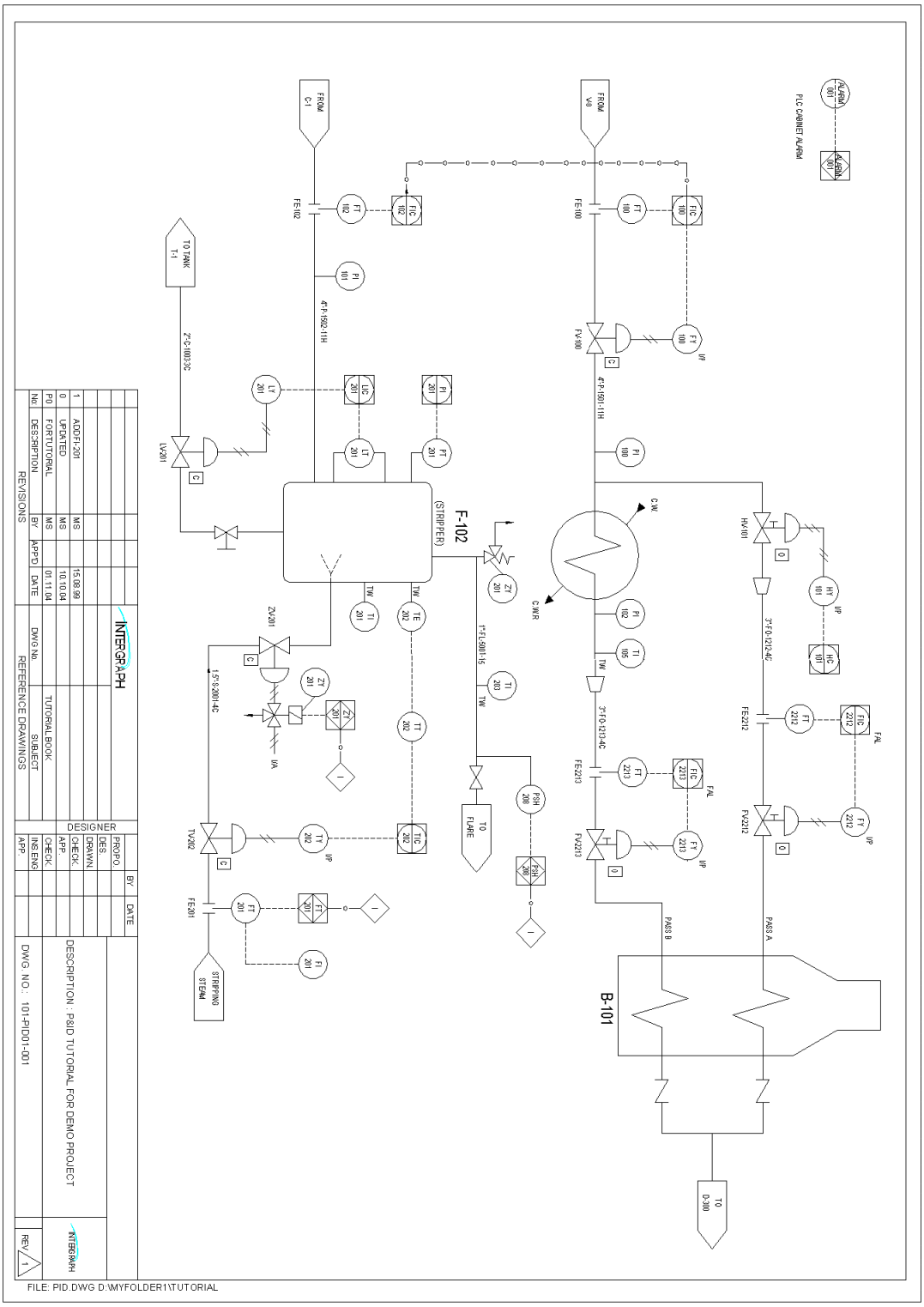
- How to view and modify a loop drawing.
- How to display additional data in enhanced SmartLoop drawings by using macros
- How to change layout settings and include annotations for enhanced reports.
- How to create block types and blocks for CAD loop drawings.
- How to associate block with instruments.
- How to add revisions.
- How to browse macros.
- How to generate a CAD loop drawing.

Generating Hook-Up Drawings

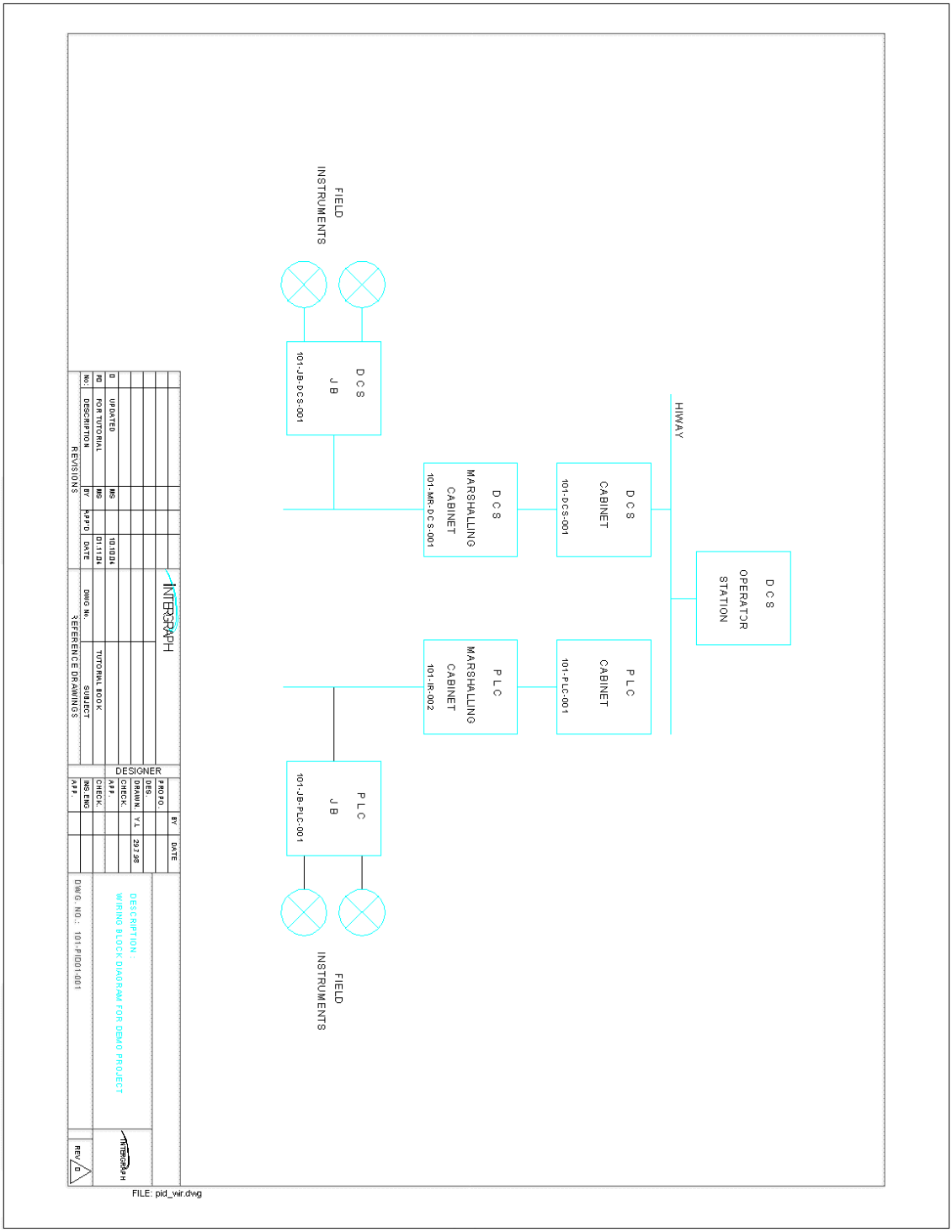
In this section, you will learn how to generate installation detail drawings for the tags that you have created, as well as Bill of Material reports. The tasks in this section include:

- How to create hook-up types and hook-ups.
- How to add items in a hook-up library.
- How to assign hook-up items to hook-ups.
- How to assign instruments to hook-ups.
- How to generate a Bill of Material.
- How to generate a hook-up drawing using the Enhanced Report Utility.

Demo P&ID


F-102
(STRIPPER)
B-101
FROM C-1
FROM V-8
TO TANK T-1
TO FLARE
TO D-300
STRIPPING STEAM
PLC CABINET ALARM
C.W.I
C.W.R
PASS B
PASS A
REVISIONS
REFERENCE DRAWINGS
DESIGNER
DESCRIPTION : P&ID TUTORIAL FOR DEMO PROJECT
DWG. NO. : 101-PID01-001
FILE: PID.DWG D:\MYFOLDER1\TUTORIAL

Wiring Block Diagram



SECTION 2

Administration Tasks

NOTE If you have already performed the system setup and configuration or if you are interested only in the instrument engineering options, you can skip directly to the Instrument Engineering options. Otherwise, proceed with the System Administration options.

The System Administration procedures are performed right after the installation of SmartPlant Instrumentation (and configuring the database by your Oracle or SQL Server database administrator). A SmartPlant Instrumentation database comes shipped with the System Administrator user definitions so that this user can log on to the Administration module and perform procedures that involve a domain initialization (creation) and configuration. In SmartPlant Instrumentation, the term **domain** most closely corresponds to a site.

The SmartPlant Instrumentation database contains several schemas. In this tutorial, you need to know about the Admin schema and Domain schema:

- The Admin schema contains the tables that enable the System Administrator to create a domain. This schema is created automatically and does not hold any SmartPlant Instrumentation data.
- The Domain schema is the schema that contains tables where users define and manage data. When initializing a domain, the System Administrator is actually creating the Domain schema in the database and populates this schema with tables.

There are two special types of users who are responsible for administration activities:

- The System Administrator is a user who works in SmartPlant Instrumentation at the Admin schema level. This means that this user can create and manage domains, set up various domain definitions, create users and define one or more users as Domain Administrators.
- A user defined as Domain Administrator can access a specific domain and then perform various activities at the Domain schema level before engineers can start entering data into the database.

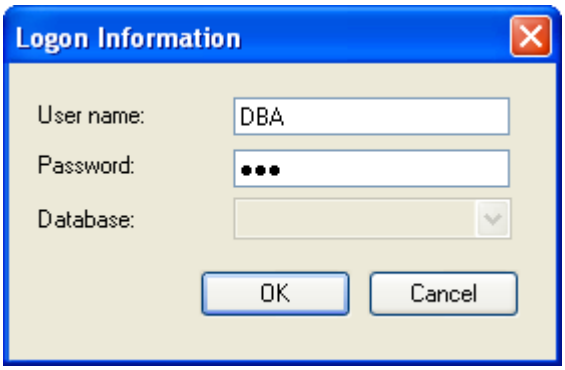
System Administrator Activities

The System Administrator activities described in this tutorial involve creating an empty domain, defining several users and assigning one of these users to the domain so that this user becomes the current Domain Administrator. Also, you will generate a number of reports showing the domain data that you defined.

Task 1 — Initializing a Domain in the Database

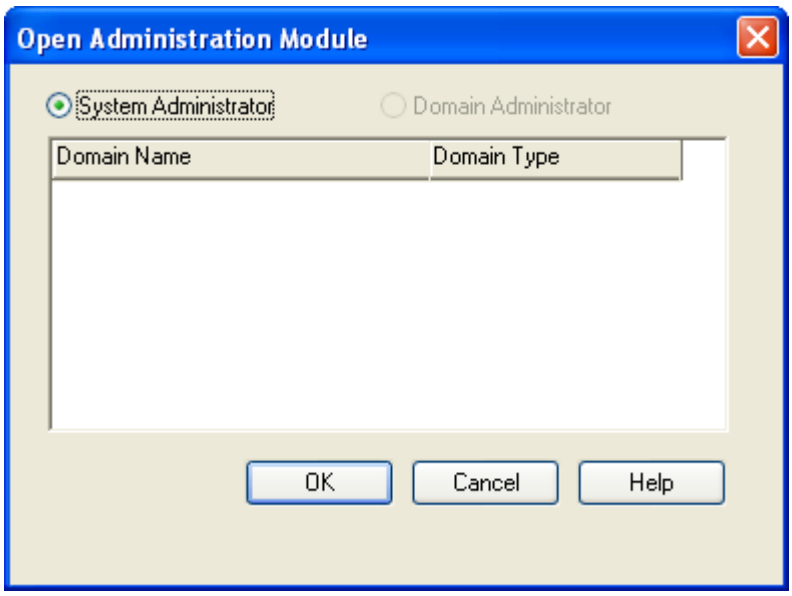
Log on as System Administrator

1. Start the Administration module as follows:
 a. On the Windows Start menu, click **Programs** > **Intergraph SmartPlant Instrumentation** > **Administration**.
 b. On the **Logon Information** dialog box, enter **DBA** as your user name and password.



TIP When you enter the system for the first time, the default user name and password are both **DBA**. The user name always appears in upper-case characters, regardless of the keyboard settings and the password is displayed masked.

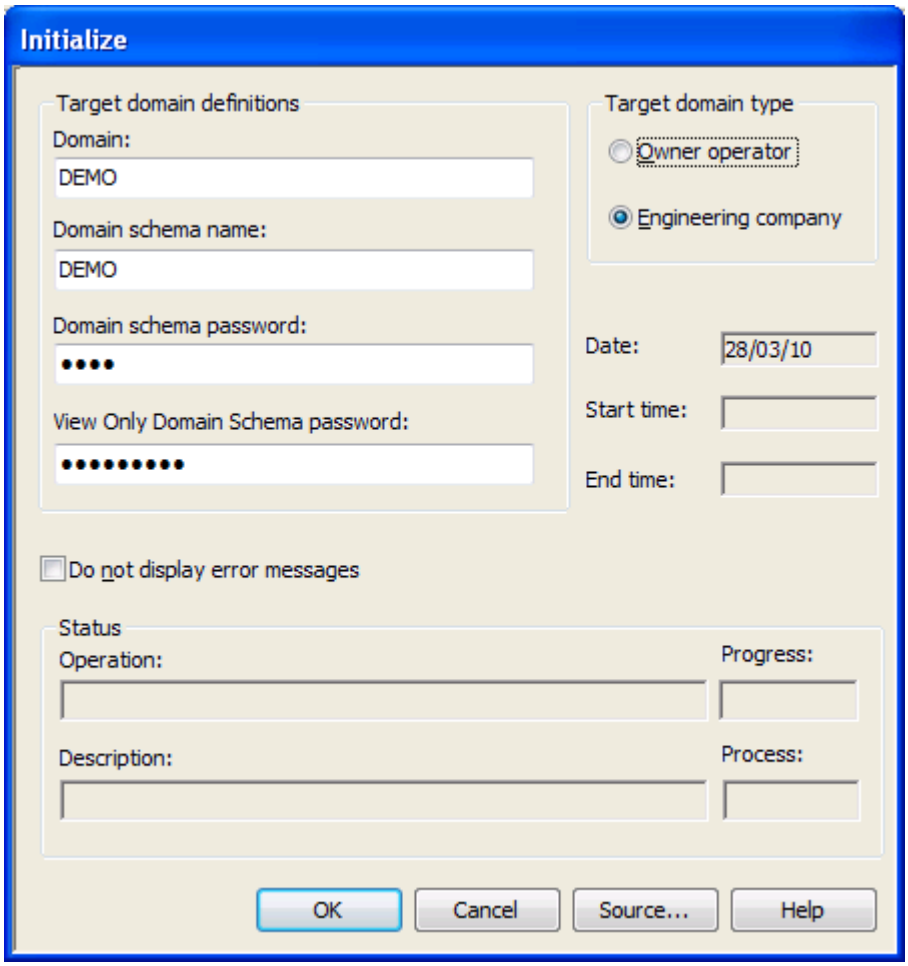
 c. Click **OK** to start the Administration module.
2. On the **Open Administration Module** dialog box, click **OK**.



NOTE The **Domain Administrator** option is not available because you have not yet defined any domains in your database.

Initialize a Domain

1. Click **File** > **Initialize**.
2. On the **Initialize** dialog box, enter data as shown.



TIPS

- Ensure that in the **Target domain type** group box, you select **Engineering company**.
- You may enter your own domain and schema names if you wish. If you are initializing an Oracle or SQL Server domain, there are additional options on this dialog box.
- Type **DEMO** in the **Domain schema password** box.
- You do not need to type a value in the **View Only Domain Schema Password** box. The view-only domain schema holds database views of all tables in a domain.
- The software displays a mask (asterisks) in the password boxes.

3. Click **OK** and wait till the process is complete.

 TIP The process may take about ten minutes, depending on your computer resources. On completion, the **Close** button appears.

4. On completion of the initialization, click **Close**.

After completing the domain initialization, there are a number of activities that you need to carry out as the System Administrator. These activities are described in the following sections.

Task 2 — Defining SmartPlant Instrumentation Users and the Domain Administrator

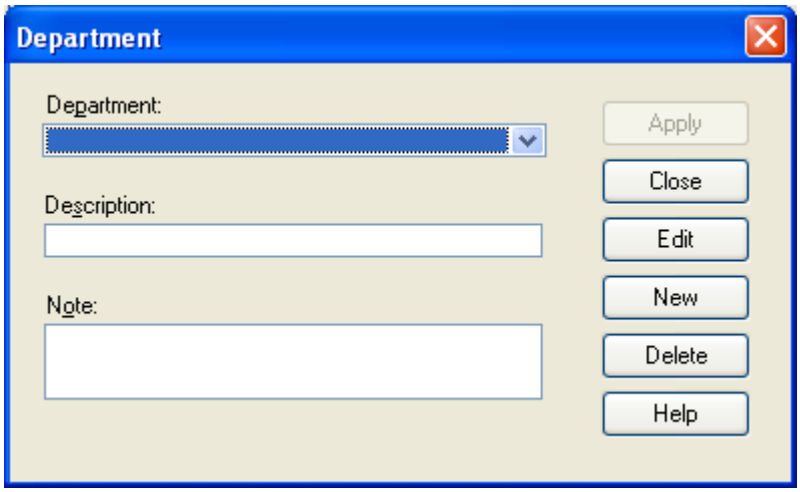
Before creating users, you must create at least one user department to which you can then assign users.

NOTE If you do not have the Administration module running in the System Administration mode, first open the module. For details, see *Log on as System Administrator* (on page 18).

Create a New Department

This activity is a prerequisite for adding users to the database.

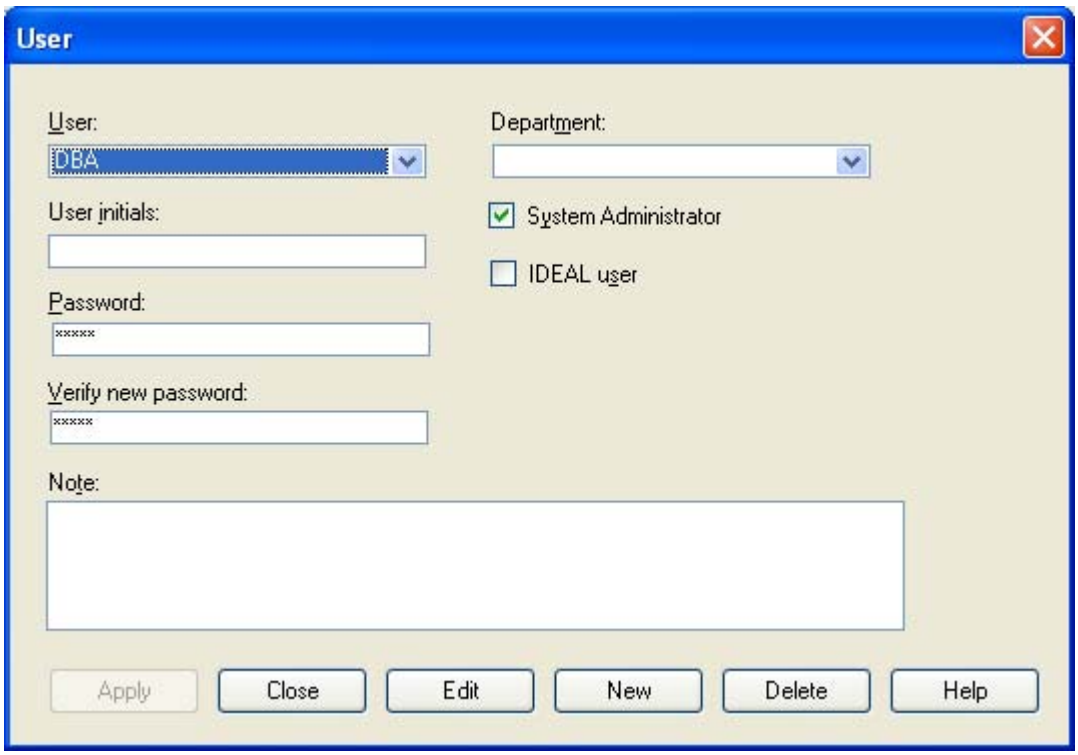
1. On the main toolbar, click to open the **Domain Definition** window.
2. From the **Domain** list, select **DEMO**.
3. On the main toolbar, click to open the **Department** dialog box.



4. Enter data as follows:
 a. Click **Edit**.
 b. In the **Department** box, type **Administration**.
 c. Click **Apply**.
 d. Click **New**.
 e. In the **Department** box, type **Instrumentation**.
 f. Type a description and a note as you require.
5. Click **Apply** and then **Close**.

Add New Users

1. Make sure that the **Domain Definition** window is open and that **DEMO** is selected in the **Domain** box.
2. Click  to open the **User** dialog box.



3. Enter data as follows:
 a. Click **New**.
 b. In the **User** box, type **USER1** (it is displayed in upper-case characters automatically).
 c. In the **User initials** box, type **U1** (using upper-case characters). These initials appear in various documents that you generate in SmartPlant Instrumentation.
 d. From the **Department** list, select **Instrumentation** to assign the new user to this department.
 e. In the **Password** box, type **USER1** as the user password (using upper-case characters).

 TIP The password is displayed masked.

 NOTE If you are using an **MSSQL** database, the password cannot match the user name.
 f. In the **Verify new password** box, type **USER1** again.
 g. In the **Note** box, type a note if required.

 TIPS

 - The **System Administrator** check box is used to grant System Administrator rights to a new user. Usually, this check box is cleared.
 - It is advisable that you keep a note of the password for each user.
4. Click **Apply**.

5. Add more users to the **Instrumentation** department so that the completed user information appears as shown.

User	Initials	Password
USER1	U1	USER1
DOMAIN	DA	DOMAIN
USER2	U2	USER2

6. Click **Close** to close the **User** dialog box.

Assign a User as a Domain Administrator

After defining all the users and assigning them to groups, you have to select a user to be assigned as the Domain Administrator who will handle all the internal configuration of the Domain.

NOTE If you do not have the Administration module running in the System Administration mode, first open the module. For details, see *Log on as System Administrator* (on page 18).

1. Click to open the **Domain Definition** window.
2. From the **Domain** list, select **DEMO**.
3. On the **Domain Definition** toolbar, click (**Edit**).
4. From the **Administrator** list, select user **DOMAIN** as Domain Administrator.

 TIP As System Administrator, you also may edit the domain name, number and description if desired.
5. Under **Domain features**, select the **Activity tracking** and **Audit trail options** check boxes.
6. Under **Specification title block**, select **Standard (used in all modules)** as the custom title block assignment method.
7. On the toolbar, click and then click .

Generate System Administration Reports

1. Log on as the System Administrator.
2. On the **Reports** menu, generate each of the available reports.
3. When done, on the **File** menu, click **Exit** to close the Administration module.

This concludes the System Administration activities required for your domain configuration. The Domain Administrator is responsible for performing subsequent administration tasks.

Domain Administrator Activities

After the System Administrator initializes a domain and defines users, the Domain schema is not ready yet to hold data. Before users can start working with SmartPlant Instrumentation, the Domain Administrator must create user groups, assign users to these groups, define access rights, and create plant hierarchy items.

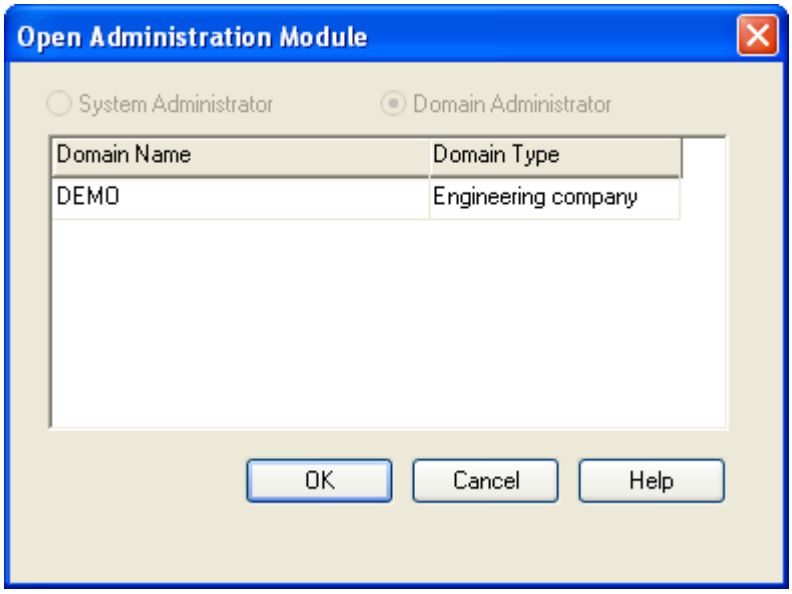
SmartPlant Instrumentation allows various groups of engineers to define and manage data without interfering with each other's activities, or holding up or overwriting each other's work. Therefore, the Domain Administrator must first create several user groups and assign users accordingly. After that, the Domain Administrator grants individual access rights for each group so that certain features of SmartPlant Instrumentation are available for users of a particular group but not available for users of another group.

NOTE In SmartPlant Instrumentation, user access rights are assigned at a group level. This means that if you need one user to have special access rights, you still need to create a group for that user.

Task 1 — Log on as Domain Administrator

This procedure allows you to log on as Domain Administrator and grant yourself maximum access rights for all operations in the **DEMO** domain. Later, after you assign users to groups, you can restrict their access to certain options.

1. On the Windows **Start** menu, click **Programs** > **Intergraph SmartPlant Instrumentation** > **Administration**.
2. On the **Logon Information** dialog box, type **DOMAIN** in both the user name and password boxes.
3. Click **OK** to open the **Open Administration Module** dialog box.



TIP Because you logged on as the Domain Administrator, only the **Domain Administrator** option is available on this dialog box.

4. Select the **DEMO** domain.
5. Click **OK** to open the **Domain Administration** window.

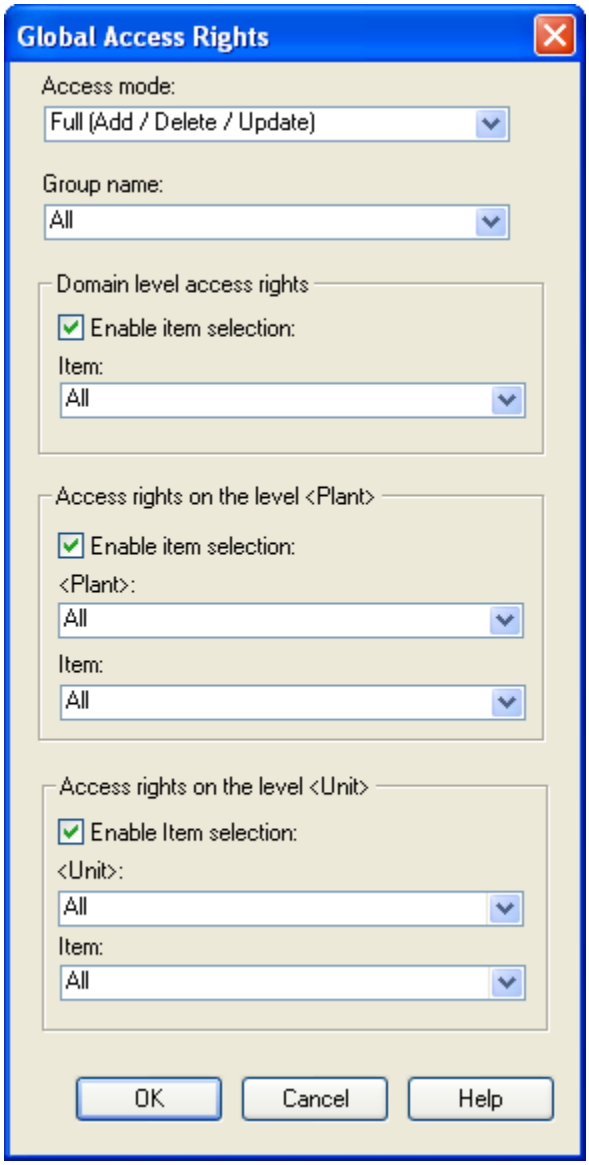
You have now successfully logged on as the Domain Administrator and you are ready to carry out the Domain Administration activities. This includes setting up the plant hierarchy (the software creates a default plant hierarchy when you define a new domain).

Task 2 — Defining Domain Administrator Access Rights

Before you can perform all of your domain administration activities, you must ensure that you have appropriate access rights as Domain Administrator.

Define Domain Administrator Access Rights

1. Click to open the **Domain Definition** window.
2. Do one of the following:
 - Click .
 - On the **Options** menu, click **Access Rights**.
3. In the **Access Rights** window, click (**Global**).
4. On the **Global Access Rights** dialog box, from the **Access mode** list, select **Full (Add / Delete /Update)**.
5. From the **Group name** list, select **All**.
6. Select all the check boxes and in each of the remaining lists, select **All**.



7. Click **OK** to save your selections and close the dialog box.

8. On the toolbar, click  and then click .

Task 3 — Creating a Plant Hierarchy

The Domain Administrator needs to create a plant hierarchy because items in SmartPlant Instrumentation are organized at different hierarchy levels. For example, instrument tag numbers always belong to a plant hierarchy item at the lowest level (by default, this is a specific unit), whereas wiring panels belong to the plant hierarchy item at the highest level in the domain (by default, this is a specific plant). For the purpose of this Tutorial, you will be using the default plant hierarchy consisting of three levels — Plant, Area, and Unit.

View the Plant Hierarchy Default Settings

1. Ensure that the **Domain Administration** window is open.
2. Click **Activities** > **Plant Hierarchy** and view the default values.

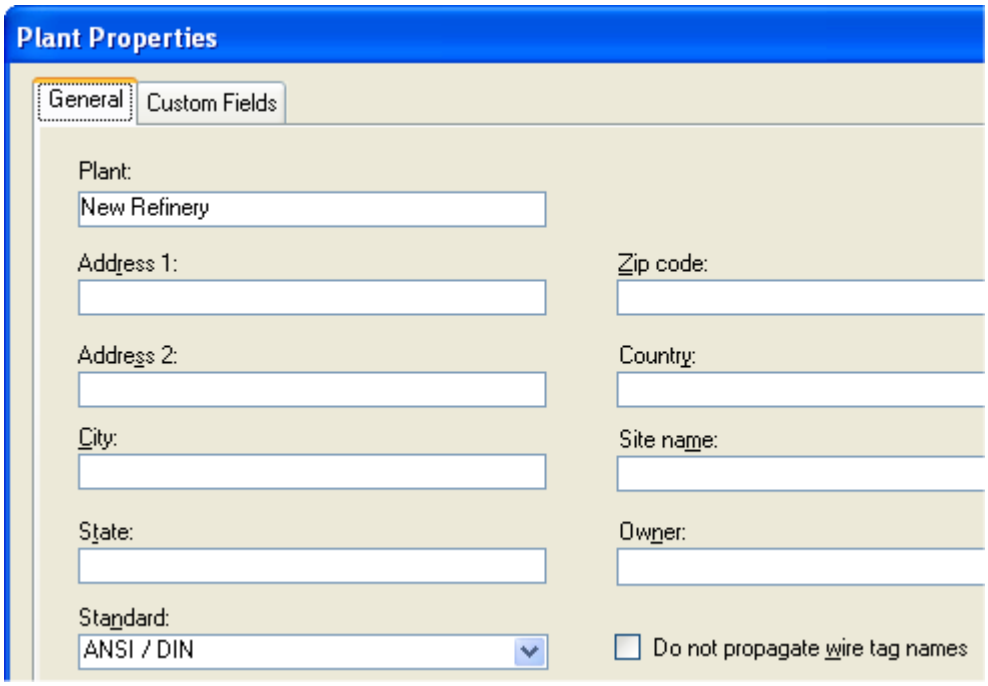
 TIP The software allows you to define a flexible hierarchy with any number of levels, for which you can specify the name of each level as you desire. Once you have created data in the plant, you are no longer allowed to modify the plant hierarchy.
3. Try changing the plant hierarchy entity names and using the **Add** and **Insert** commands to create more levels.
4. Click **Cancel** to exit without saving any changes.

Create a Plant Hierarchy

1. With the Domain Administration window open, click Activities > Plant Hierarchy Explorer.

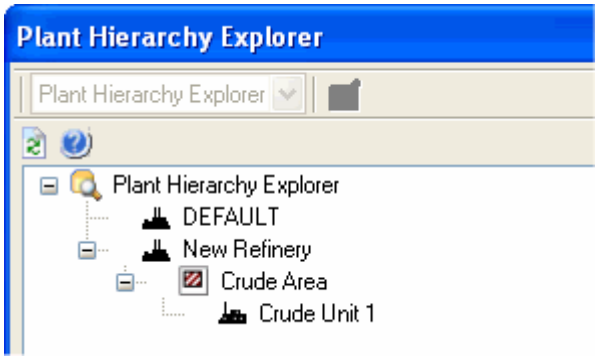
 TIP The software allows you to define a flexible hierarchy with any number of levels, for which you can specify the name of each level as you desire.
2. On the tree, right-click **Plant Hierarchy Explorer**.
3. On the shortcut menu, click **New** > **Plant**.
4. On the **Plant Properties** dialog box, do the following:
 a. In the **Plant** box, name your plant **New Refinery**.
 b. From the **Standard** list, select **ANSI / DIN** as the default pipe standard for the plant.
 c. Ensure that the **Do not propagate wire tag names** check box is cleared.

The **Plant Properties** dialog box should appear as shown:



5. Click **OK**.
6. In the **Plant Hierarchy Explorer**, right-click **New Refinery**.
7. On the shortcut menu, click **New** > **Area**.
8. On the **Area Properties** dialog box, do the following:
 a. In the **Name** box, type **Crude Area**.
 b. Click **OK** to close the dialog box.
9. In the **Plant Hierarchy Explorer**, click ⊞ beside **New Refinery** to expand the hierarchy, and then right-click **Crude Area**.
10. On the shortcut menu, click **New** > **Unit**.
11. On the **Unit Properties** dialog box, do the following:
 a. In the **Name** box, type **Crude Unit 1**.
 b. In the **Number** box, type **101**.
 c. Click **OK** to close the dialog box.
12. Click ⊞ beside **Crude Area** to expand the hierarchy.

 The **Plant Hierarchy Explorer** should now appear as shown:



13. Click  to close the **Plant Hierarchy Explorer**.

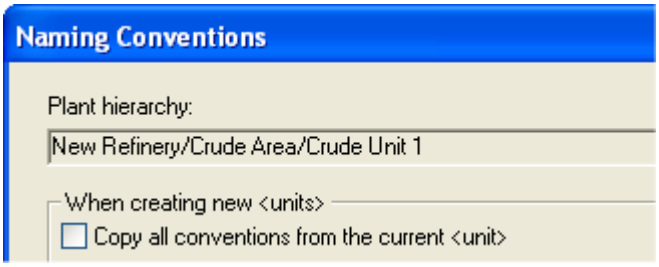
You have just completed organizing the process plant hierarchy. Now you need to define instrument tag and loop naming conventions for **Crude Unit 1**.

Task 4 — Defining Instrument Tag and Loop Naming Conventions

This task involves defining naming conventions for instruments (tag numbers) and loops in the unit in which you will be working using the ISA Standard. After defining the naming conventions, it is possible to start adding data in SmartPlant Instrumentation.

Define Instrument Tag Naming Conventions

1. With the **Domain Administration** window open, do one of the following:
 - Click .
 - Click **Activities** > **Naming Conventions** to open the **Naming Conventions** dialog box.
2. Beside **Plant hierarchy**, click **Browse** and select the unit you created. The **Plant hierarchy** box should now display the hierarchy **New Refinery/Crude Area/Crude Unit 1**:

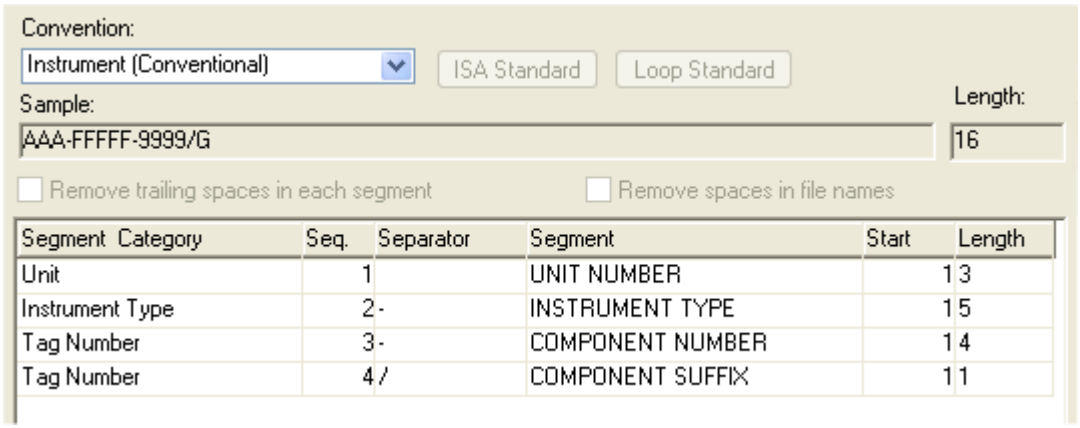


3. From the **Convention** list, select **Instrument (Conventional)**.
4. Click **ISA Standard**.

 TIP According to the ISA standard, the instrument tag string can contain up to four different segments. Before each segment, you can add separator characters in the **Separator** column. Each segment has a starting character position and length.
5. Select the following segment categories and segments:

Segment Category	Segment
Unit	UNIT NUMBER
Instrument Type	INSTRUMENT TYPE
Tag Number	COMPONENT NUMBER
Tag Number	COMPONENT SUFFIX

6. Enter separators before the second, third and fourth tag segments and modify the string lengths in the **Length** column as shown:



Segment Category	Seq.	Separator	Segment	Start	Length
Unit	1		UNIT NUMBER	1	3
Instrument Type	2	-	INSTRUMENT TYPE	1	5
Tag Number	3	-	COMPONENT NUMBER	1	4
Tag Number	4	/	COMPONENT SUFFIX	1	1

TIPS

- The **Sample** field shows the changes in the tag structure as you make your naming convention definitions.
- The first segment in the sequence becomes the tag prefix. In the current naming convention, the unit number **101** will be the prefix of every instrument you create in the current unit.
- You can only change the tag convention for a unit as long as no instruments are defined for the unit.

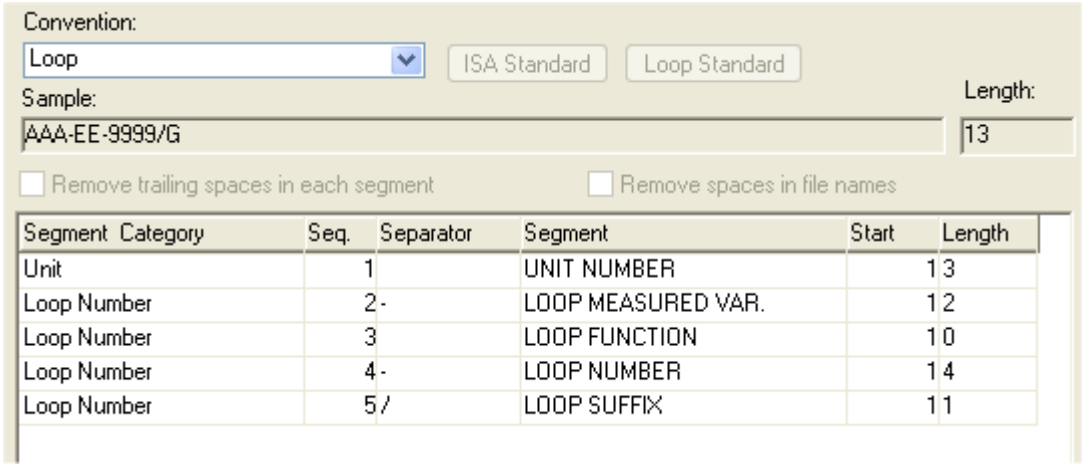
7. Click **Apply**.

Define Loop Naming Conventions

1. In the **Convention** list, select **Loop**.
2. Click **ISA Standard**.

 TIP Make sure the correct unit is selected, which is **Crude Unit 1**.

3. Define the segments, separators, and segment lengths as shown:



Segment Category	Seq.	Separator	Segment	Start	Length
Unit	1		UNIT NUMBER	1	3
Loop Number	2	-	LOOP MEASURED VAR.	1	2
Loop Number	3		LOOP FUNCTION	1	0
Loop Number	4	-	LOOP NUMBER	1	4
Loop Number	5	/	LOOP SUFFIX	1	1

4. Click **Apply** and then close the dialog box.

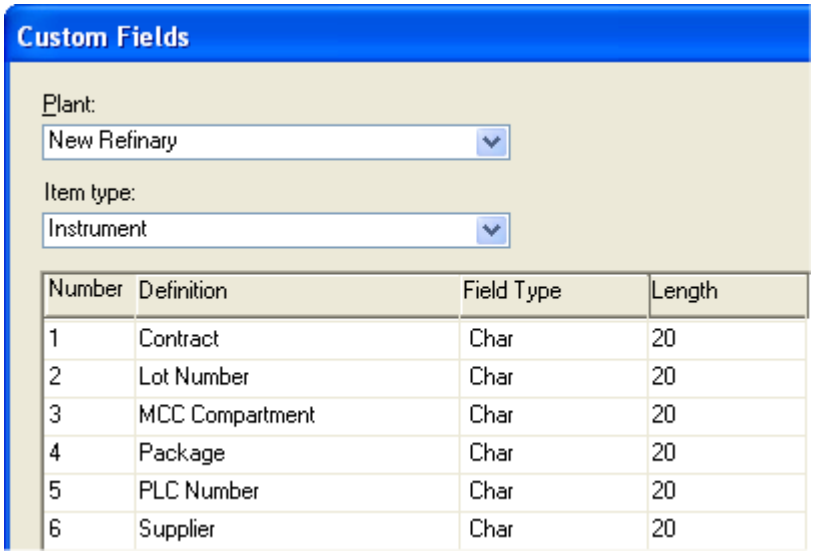
Task 5 — Creating Custom Fields and Tables

Custom fields are used when you want to add your own data values in character, numeric, or date fields. To add your own supporting tables (used to create lists), use custom tables.

Create Instrument Index Custom Fields

In SmartPlant Instrumentation, each item type contains a number of predefined property fields where you can enter data values. If needed, the Domain Administrator can add custom fields for the available item types. In this procedure, you will define custom fields for instruments. After that, when you log on as a SmartPlant Instrumentation user, you can display these fields and enter values whenever you need to work with instrument tag properties.

1. With the **Domain Administration** window open, do one of the following:
 - Click .
 - Click **Activities** > **Custom Fields**.
2. From the **Plant** list, select **New Refinery**.
3. From the **Item type** list, select **Instrument**.
4. Type definitions for fields 1 through 6 as shown.



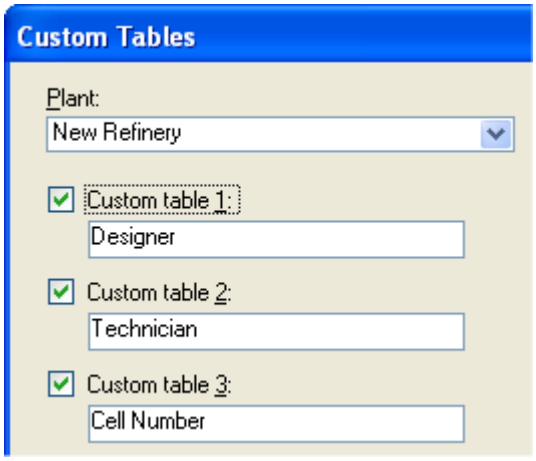
Number	Definition	Field Type	Length
1	Contract	Char	20
2	Lot Number	Char	20
3	MCC Compartment	Char	20
4	Package	Char	20
5	PLC Number	Char	20
6	Supplier	Char	20

5. Click **Apply**.
6. At the prompts, click **OK**, and then click **Close**.

Create Custom Tables

Unlike custom fields, which are defined per item type, custom tables are available as additional supporting tables of the Instrument Index module only. In the Instrument Index module, the values that you add in these tables can be assigned to instrument tags as additional instrument properties.

1. With the **Domain Administration** window open, click **Activities** > **Custom Tables**.
2. From the **Plant** list, select **New Refinery**.
3. Select the check boxes beside the first three custom tables.
4. Type a name for each of the custom tables that you selected as shown.



5. Click **Apply**.

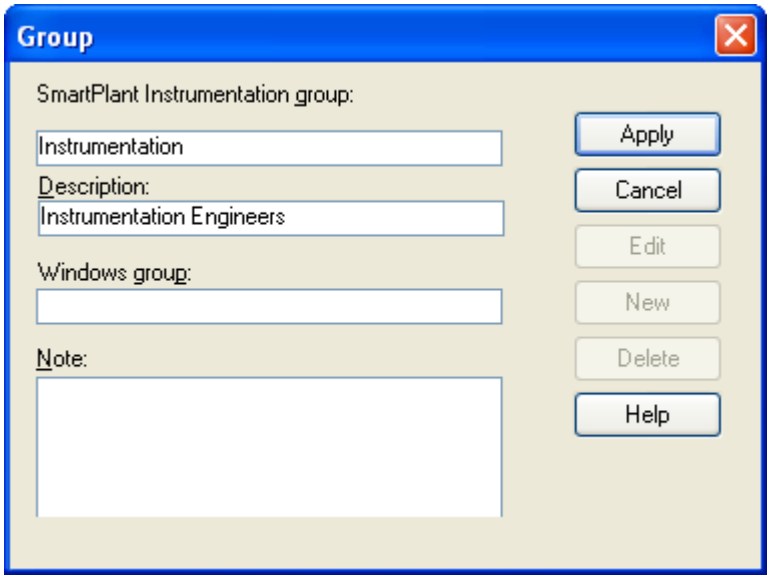
Task 6 — Organizing Users into Groups and Granting Access Rights

Currently, you have only two users who can access SmartPlant Instrumentation: the System Administrator with the **DBA** logon name and password and the Domain Administrator with the **DOMAIN** logon name and password. Other users, defined in Task 2 of System Administrator Activities, cannot log on to SmartPlant Instrumentation because they do not yet belong to any user group. Both System and Domain Administrators are assigned automatically to the ADMINISTRATORS user group, which comes shipped with SmartPlant Instrumentation.

In this task, you will assign the remaining users to a new group and define access rights for that group.

Create a User Group

1. With the **Domain Administration** window open, click **Activities** > **Group**.
2. On the **Group** dialog box, click **New**.
3. In the **SmartPlant Instrumentation group** box, type **Instrumentation**.
4. In the **Description** box, type **Instrumentation Engineers**.



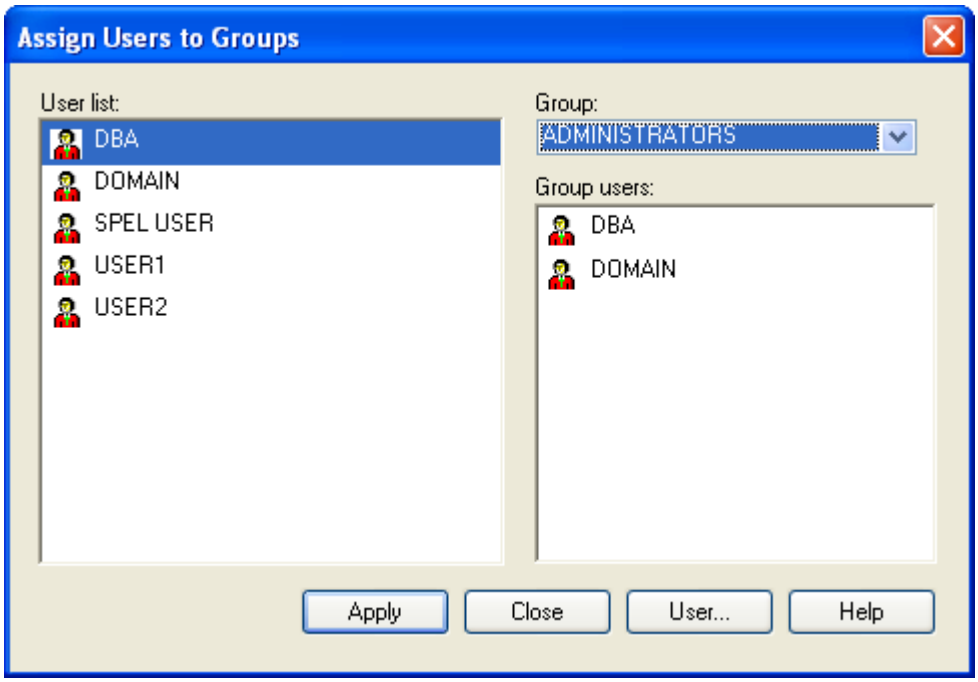
5. Click **Apply**, and when prompted whether to copy access rights from another group, click **No**.
6. Click **Close**.

Assign a User to the New Group

After creating the new group, the Domain Administrator assigns users to that group.

NOTE A group can contain several users or one user only. When a group contains one user only, you can exercise tight control or grant special access rights to one person.

1. With the **Domain Administration** window open, click **Activities** > **Assign Users to Groups** to open a dialog box where you can view the entire list of existing SmartPlant Instrumentation users.

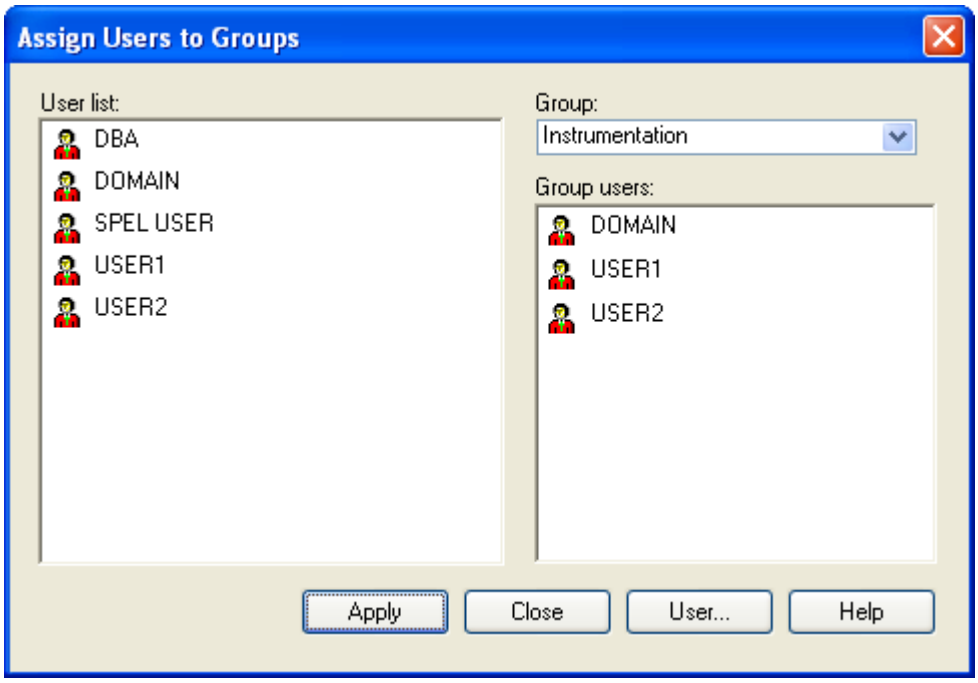


2. From the **Group** list, select **Instrumentation**.
3. Select and drag users **USER1**, **USER2**, and **DOMAIN** from the **User list** pane to the **Group users** pane.

TIPS

- If desired, you can assign the same user to more than one group.
- You can select a user and click **User** to open the **User** read-only dialog box to display a complete user description for the selected user.
- If you want to remove a user from a group, drag that user from the **Group users** pane to the **User list** pane.

The **Assign Users to Group** dialog box should now appear as shown.

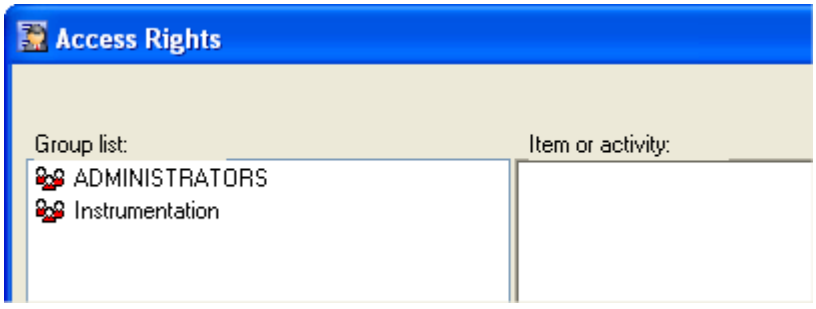


4. When done, click **Apply** and close the dialog box.

Grant Access Rights for a Group

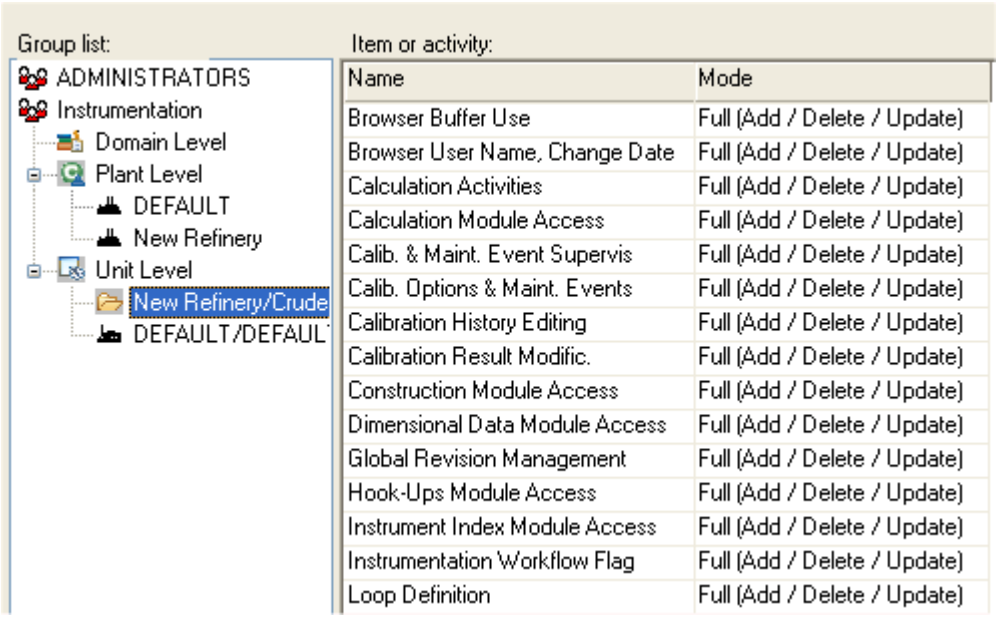
Once groups and users have been defined, the Domain Administrator grants them access rights to various activities. Access rights are granted **per group**, which means that all users in the selected group are granted the same access rights.

1. In the **Domain Administration** window, click to open the **Domain Definition** window.
2. On the **Domain Definition** toolbar, click to open the **Access Rights** window with the user group displayed in the **Group list** section of the window.

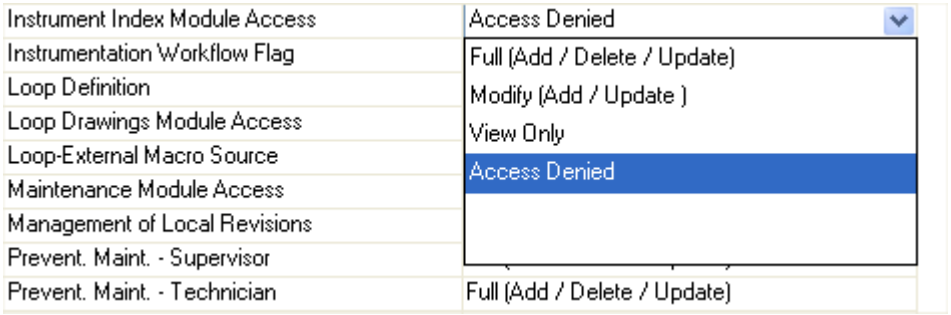


3. Double-click **Instrumentation** to display the access levels for that group.
4. Expand **Unit Level** by clicking the ⊞ icon to display all the units in the domain.
5. Select the unit — **New Refinery/Crude Area/Crude Unit 1**.

The **Item or activity** section of the window displays the list of items and activities that apply at the unit level:



6. From the **Item or activity** column, scroll down the **Name** list to **Instrument Index Module Access**.
7. From the **Mode** list, select **Access Denied**:

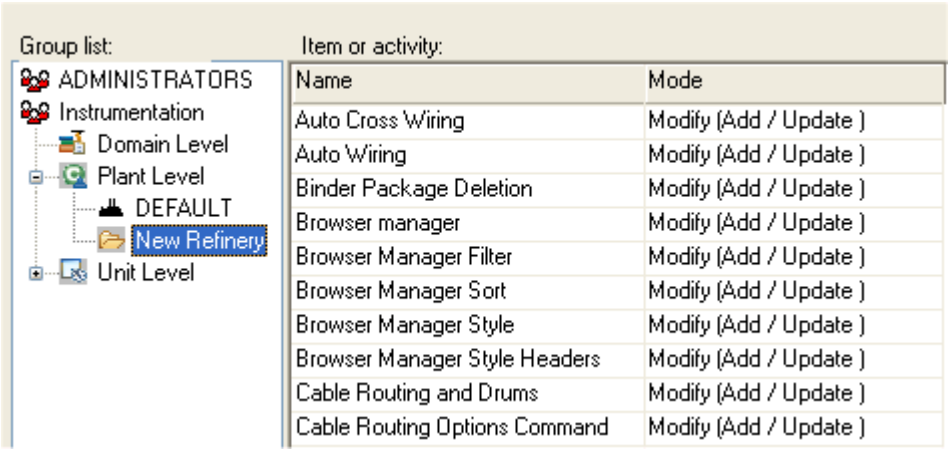


8. On the toolbar, click (**Save**) and close the window.
9. Test your access rights as follows:
 a. Log on to SmartPlant Instrumentation as **USER1**.
 b. On the **Open** dialog box, select **New Refinery** > **Crude Area** > **Crude Unit 1**.
 c. Click the **Modules** menu and note that all modules are available except for the Instrument Index module.
 d. Exit SmartPlant Instrumentation.
10. In the Administration module, reopen the **Access Rights** window and reset **Instrument Index Module Access** to **Full (Add / Delete /Update)**.
11. On the toolbar, click to close the **Access Rights** window; and click **Yes** when prompted to save changes.

Apply Access Rights for More than One Group or Item

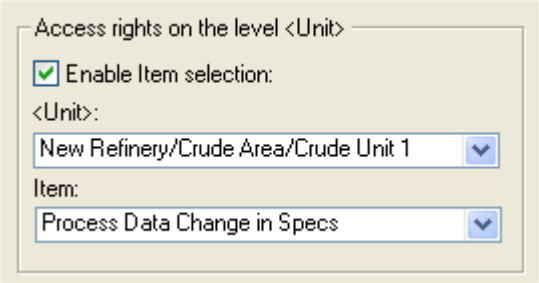
You can grant the same access rights for all the user groups in the domain or for all the items at each level (for a particular group or for all groups). You have already used this option to grant full access to the Domain Administrator for all items. You will now modify access rights globally for the **Instrumentation** group and perform other exercises to learn how this option works.

1. In the **Domain Administration** window, click to open the **Domain Definition** window.
2. On the **Domain Definition** toolbar, click to open the **Access Rights** window.
3. Click to open the **Global Access Rights** dialog box.
4. From the **Access mode** list, select **Modify (Add / Update)**.
5. From the **Group name** list, select **Instrumentation**.
6. Define access rights at the plant level as follows:
 a. Under **Access rights on the level <Plant>**, select the **Enable item selection** check box.
 b. In the **Plant** list, select **New Refinery**.
 c. In the **Item** list, select **All**.
 d. Click **OK**.
7. On the **Access Rights** dialog box, do the following:
 a. Double-click the **Instrumentation** group to display the access levels.
 b. Expand **Plant Level** by clicking the ⊞ icon to display all the plants in the domain.
 c. Click **New Refinery**.
 d. In the **Mode** column of the right pane, ensure that all the items for the New Refinery plant are defined as **Modify (Add / Update)**.



8. Click to reopen the **Global Access Rights** dialog box.
9. From the **Access mode** list, select **Access Denied**.
10. From the **Group name** list, select **All**.
11. Define access rights at the unit level as follows:
 a. Under Access rights on the level <Unit>, select the **Enable item selection** check box.
 b. From the <Unit> list, select **New Refinery/Crude Area/Crude Unit 1**.

c. From the **Item** list, select **Process Data Change in Specs** as shown:



12. Click **OK**.
13. In the **Access Rights** window, check the results as follows:
 a. Double-click **Instrumentation** to display the access levels for that group.
 b. Expand **Unit Level** by clicking the ⊞ icon to display all the units in the domain.
 c. Click New Refinery/Crude Area/Crude Unit 1.

 In the **Mode** column of the right pane, the **Process Data Change in Specs** item for **Crude Unit 1** should be defined as **Access Denied**. The same condition should apply for this unit in the **ADMINISTRATORS** group.
14. Click  to save your selection to the database.

 CAUTION At the end of this exercise, make sure that all the items are returned to **Full (Add / Delete / Update)** access.
15. When done, click  to close the **Access Rights** window and return to the **Domain Definition** window.

Task 7 — Setting Report Management Options

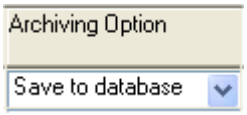
The Domain Administrator can define various report settings and revision archiving options, and if custom title blocks have been created, apply them to reports. In the following exercise, you will define report archiving options.

Define Archiving Options for Reports

You must specify archiving options for all those reports for which you want to use the report comparison feature in SmartPlant Instrumentation.

1. On the **Domain Administration** window menu bar, click **Activities** > **Report Management**.
2. Under **Filter by**, filter the list of reports by selecting **Instrument Index** from the **Module** list, and select the **Apply** check box.
3. From the list of reports, under the **Archiving Option** column, select **Save to database** for the following reports:
 - **All Tag Numbers Report – All <Units>**
 - **All Tag Numbers Report – Current <Area>**
 - **All Tag Numbers Report – Current <Unit>**
 - **Instrument Statuses Report**

 The **Archiving Option** column for these four reports should display **Save to database**:



4. Under **Filter by**, filter the list of reports by selecting **Specifications** from the **Module** list, and select the **Apply** check box.
5. For the **Specification Sheets** report, click the **Save Document Data** check box and under the **Archiving Option** column, select **Save to database**.

 TIP The **Save Document Data** option saves all revisions for the report so that you can use the **Changed Documents** feature (accessed from the **Tools** menu in SmartPlant Instrumentation). This option does not affect report comparison, which is made available by the selection under **Archiving Option**.
6. Click **OK**.

Task 8 — Performing Miscellaneous Activities

The following procedures are optional. Completion of some of these procedures affects only specific scenarios you perform in SmartPlant Instrumentation. Therefore, you can already start working with SmartPlant Instrumentation and return to these tasks later, if needed.

Assign a Domain Logo

You need to perform this procedure if you want to display your company logo in report title blocks.

1. With the **Domain Administration** window open, click  to open the **Domain Definition** window.
2. On the **Domain Definition** toolbar, click .

 The **Select Logo** dialog box appears as shown:



3. Click **Browse** to open the **Select a Logo File** dialog box.
4. Select a .bmp file and click **Open** to display the in the **Logo preview** area of the **Select Logo** dialog box.
5. Click **OK** to assign the displayed logo to all documents that users can generate in the current domain.

Generate Domain Administrator Reports

As Domain Administrator, you can generate a number of reports concerning Domain Administration. For example, the Domain Statistics report shows a summary of the shipped reference data items that are included in the database.

1. With the **Domain Administration** window open, click **Reports** > **Domain Statistics**.
2. When prompted to preview the report, click **Yes**.

 TIP The non-zero values for some of the items represent shipped data.
3. Click ▶ to scroll forward through the pages of the report.
4. When done, on the **File** menu, click **Exit** to exit the Administration module.

This completes the Domain Administration activities required for your domain configuration. You will perform all subsequent tasks in SmartPlant Instrumentation.

SECTION 3

Getting Started with SmartPlant Instrumentation

NOTES

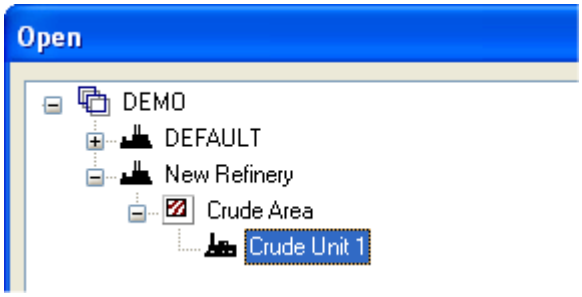
- Before starting SmartPlant Instrumentation, ensure that the Domain Administrator has created a unit and a user group, and defined the naming conventions for the unit and access rights for the group.
- If find that you are unable to perform certain tasks, make sure that the Domain Administrator has granted you appropriate access rights.

Task 1 — Logging on to SmartPlant Instrumentation

For the plant hierarchy example that you have created for this Tutorial, the lowest plant hierarchy level is UNIT. When you log on to SmartPlant Instrumentation, you must select the unit in which you are going to work.

1. On the Windows **Start** menu, click **Programs** > **Intergraph SmartPlant Instrumentation** > **SmartPlant Instrumentation**.
2. On the **Logon Information** dialog box, type **USER1** as both your user name and password.

 NOTE Upon first login you will be offered to change the password set by the system administrator. Click **Yes** to change the password or **No** to disregard. You will not be prompted during subsequent logins.
3. On the **Open** dialog box, expand the **DEMO** domain hierarchy and select **Crude Unit 1** as shown:



4. Click **OK** to open **Crude Unit 1** in SmartPlant Instrumentation.

Task 2 — Defining Default SmartPlant Instrumentation Properties

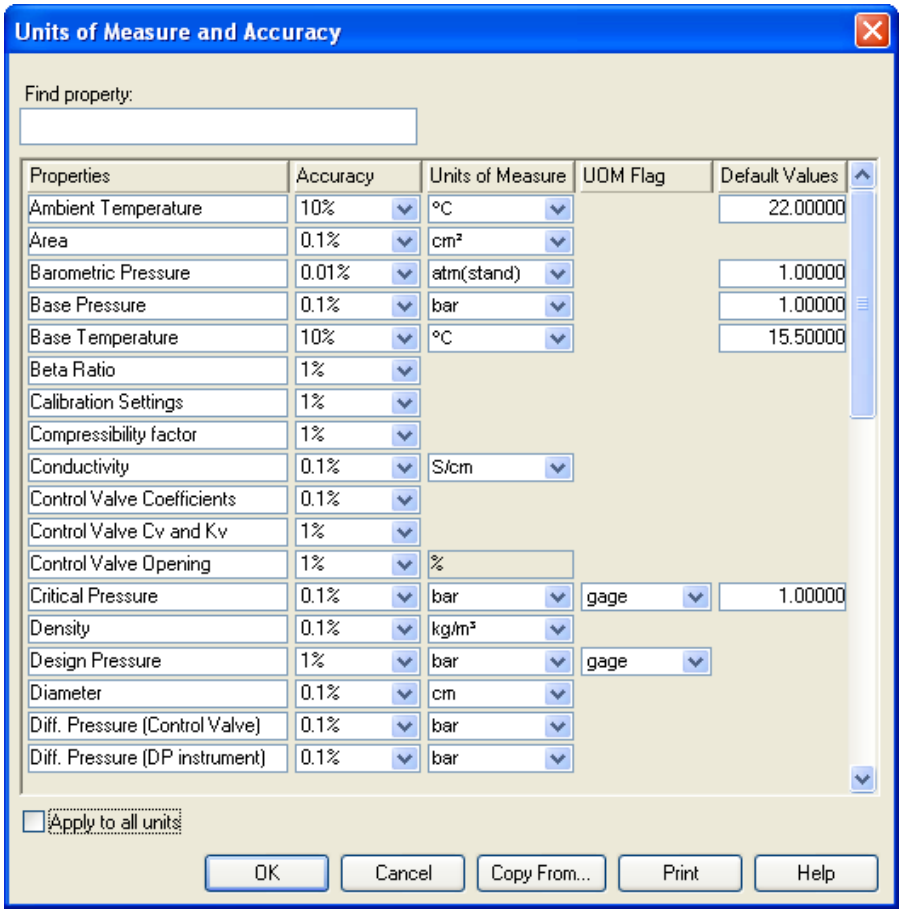
In the following tasks, you are going to define default values for units of measure and accuracy, and set some user preferences.

Define Units of Measure and Accuracy

This option allows you to specify the display accuracy for properties that appear in spec sheets, process data and calculation sheets, and calibration data sheets. You can also set default units of measure where appropriate and, in some cases, specify default values.

NOTE The accuracy settings affect the display only and do not influence the accuracy of the values used for calculations.

1. Click **File** > **Units of Measure and Accuracy**.
2. On the **Units and Measure and Accuracy** dialog box, enter values as shown in the example.



3. Click **OK** to accept the values and close the dialog box.

NOTE The units of measure and accuracy values that you entered are **defaults**. If, when performing a task later on, you find that the units of measure or display accuracy in the software do not match those shown in the Tutorial, you can override these values where they appear, for example, on a specification sheet.

Set SmartPlant Instrumentation Preferences

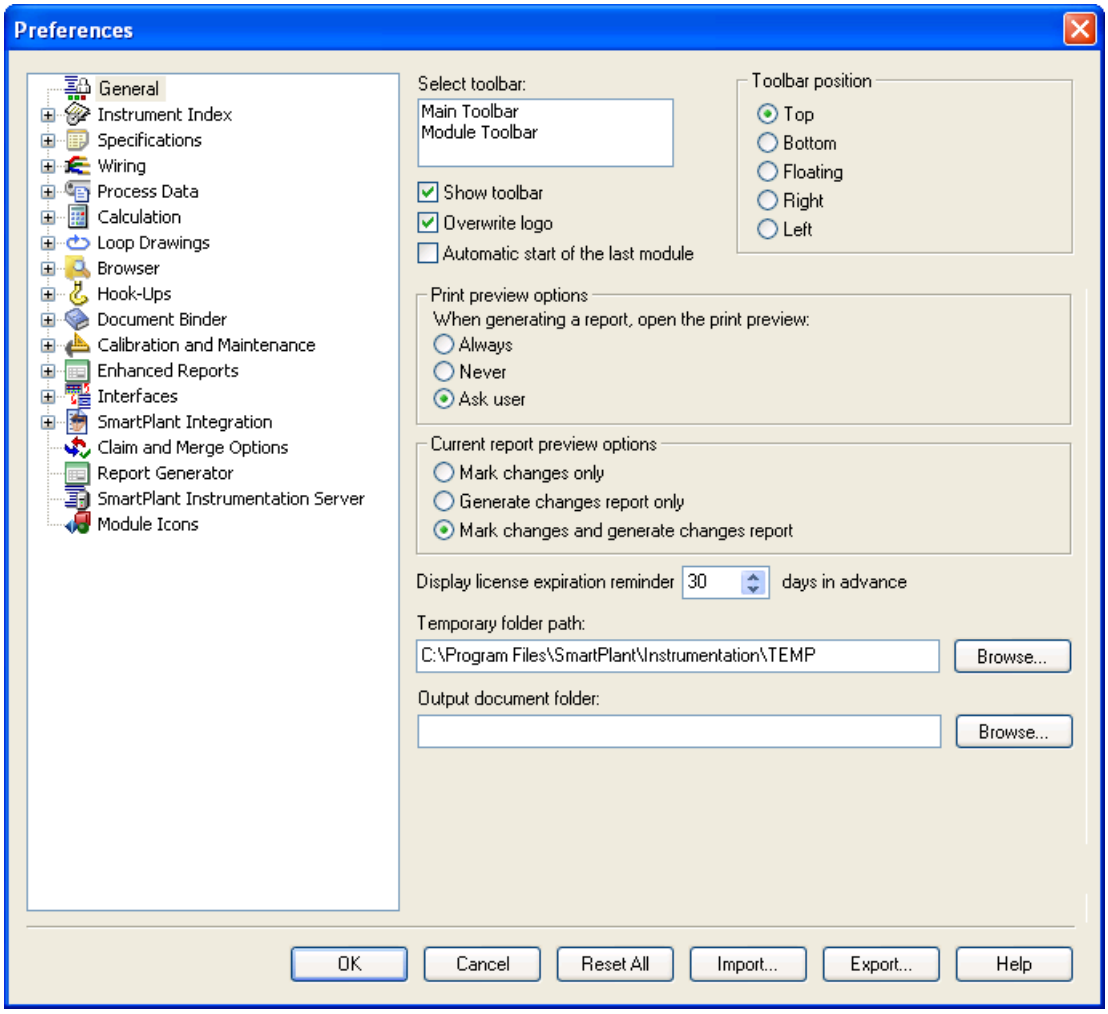
Since preferences apply per user, this option allows you to specify personal settings that will apply to your computer only.

NOTES

- If you are a new user, the default values for the preferences settings are those that are specified by the Domain Administrator on the **Preferences Management** dialog box.
- If the Domain Administrator has modified the advanced preferences management settings, some of the options on the **Preferences** dialog box may not be available to you.

1. Click **File** > **Preferences**.
2. In the tree view, click **General**.
3. Beside **Temporary folder path**, click **Browse** to navigate to a suitable location (for example, the TEMP folder under the folder where you installed SmartPlant Instrumentation).

 The page should now appear as shown:



4. In the tree view, click **Specifications**.
5. Click **General**, and select **Allowed** under **Tag creation**.
6. Click **Multi-Tag Specifications**, and tick **Show master tag** check box.

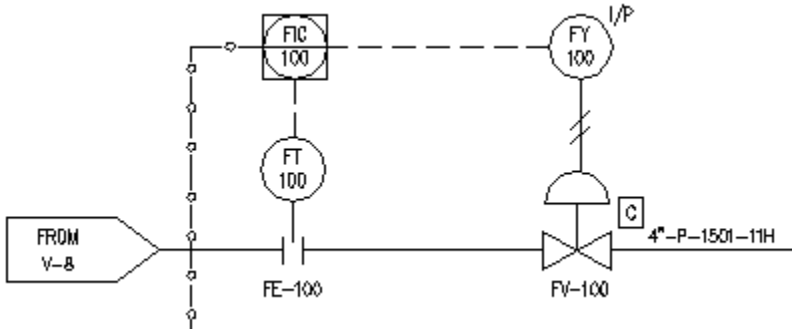
TIP The first tag that you add in a multi-tag spec becomes the master tag. The spec pages upon which the master tag is based become the common pages for all the tags in the specification. You can change master tags within a multi-tag specification. After selecting the **Show master tag** check box in the common data page of your multi-tag spec, the software indicates the master tag number in the **Tag Number** column.

7. Click **OK** to save the modified preferences.

SECTION 4

Creating Instruments and Control Loops

You can perform most of the operations needed to create instruments and control loops from the **Domain Explorer**. The following diagram shows a line with one of the control loops and its instruments that you will be creating in this Tutorial.

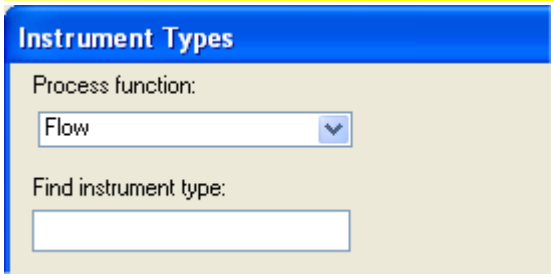


Task 1 — Defining Instrument Type Profiles

Instruments are classified according to their process function and instrument type. Every instrument that you create in SmartPlant Instrumentation has a process function and must be assigned to an instrument type. Examples of process functions are level, pressure, temperature, control valve, relief valve, and so forth. There is also a process function named **General** which is used for instruments that are not intended for one particular process. Instruments of a particular type usually have predefined properties or items, which you specify in the instrument type profile; for example, you can specify that when creating any instrument of a particular type, the software also creates a specification sheet, a device panel, and a cable. In the definitions used in this Tutorial, the instrument type also comprises part of the tag naming convention.

Define Instrument Types

1. Start the Instrument Index module by doing one of the following:
 - On the main toolbar, click .
 - On the main menu bar, click **Modules** > **Instrument Index**.
2. Click **Tables** > **Instrument Types**.
3. On the **Instrument Types** dialog box, from the **Process function** list, select **Flow**.



4. Create a new instrument type as follows:

a. Click **New** to add a new row.
b. Type the instrument type name **FE**.
c. Press the **Tab** key and type the description **D/P Type Flow Element**.
d. Press the **Tab** key again and type **FE** in the **CS Tag Instrument Type Alias** column.
e. Click **Apply**.

5. Additionally, define all the instrument types that appear in the following table:

Process Function	Instrument Type	Description
Flow	FT	D/P Type Flow Transmitter
General	FY	I/P Transducer (flow)
General	TY	I/P Transducer (temperature

6. For the **FI** instrument type (**Flow** process function), rename the description **FLOW INDICATOR** to **Local Flow Indicator**.

NOTES

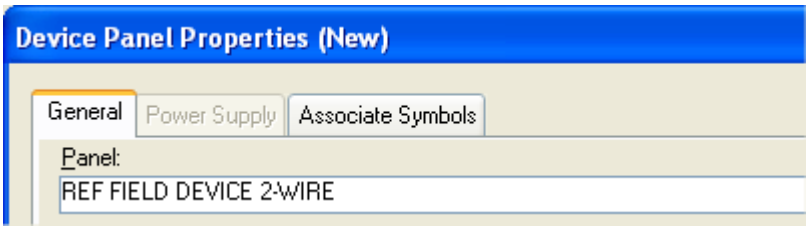
- The **Description** field is required as part of the instrument type definition. You can have more than one instrument type with the same name, provided the descriptions are unique. Conversely, different instrument types can have the same description as long as overall the combination of the instrument type and description is unique.
- Other instrument types that are used in the Tutorial come shipped with SmartPlant Instrumentation.

7. When done, click **OK** to accept the new values and close the dialog box.

Define Reference Wiring for a Field Device

To enable automatic creation of device panels when creating instrument tags, you must first define appropriate reference device panels and cables in the **Reference Explorer** and then associate these panels and cables with the desired instrument type profiles.

1. Click **Tools** > **Reference Explorer**.
2. Click the **Panels** folder to expand it and right-click the **Device Panels** folder.
3. On the shortcut menu, click **New** > **Device Panel (Conventional)**.
4. On the **Device Panel Properties** dialog box, type the information as shown and click **OK**.



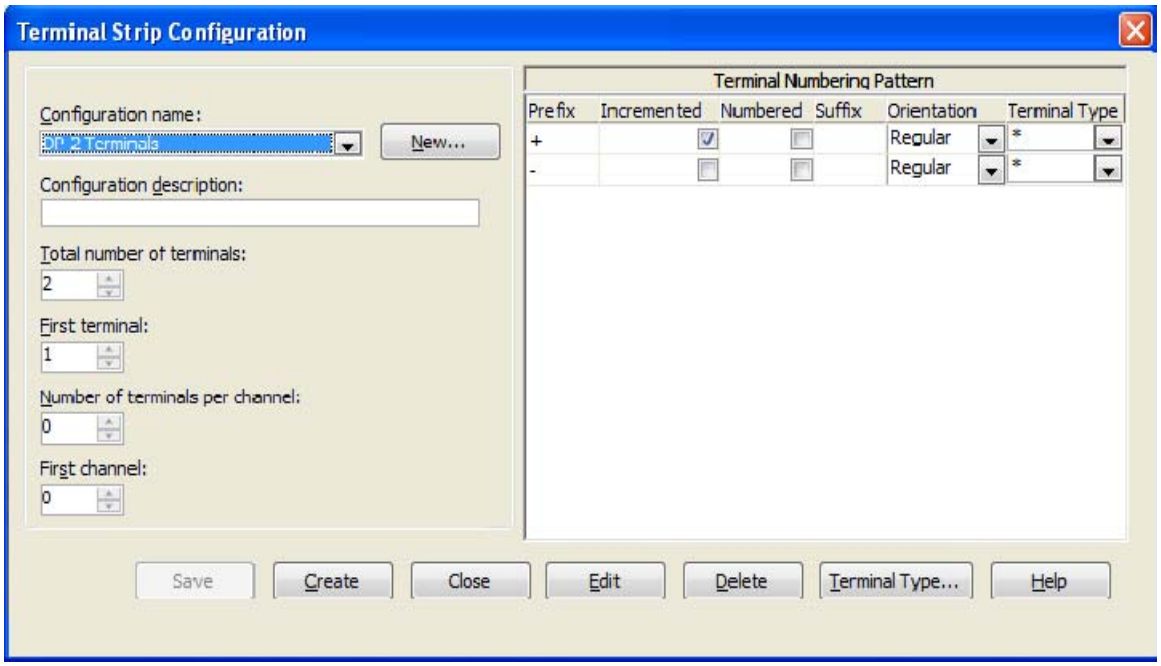
5. In the **Reference Explorer**, click the ⊞ icon to expand the **Device Panels** folder.
6. Right-click the **REF FIELD DEVICE 2-WIRE** panel.
7. Create a strip and terminals for this device panel as follows:
 a. On the shortcut menu, click **New** > **Terminal Strip**.
 b. On the **Terminal Strip Configuration** dialog box, beside the **Configuration name** box, click **New**.
 c. On the **Number of Terminals in Pattern** dialog box, enter **2** as the number of terminals in the pattern, and then click **OK** to return to the **Terminal Strip Configuration** dialog box.

d. In the **Configuration name** box, type **DP 2 Terminals**.

e. Under **Terminal Numbering**, in the **Prefix** column, type **+** and **-**.

f. Select the **Incremented** check box for the **+** terminal.

The values in this dialog box should now appear as shown:



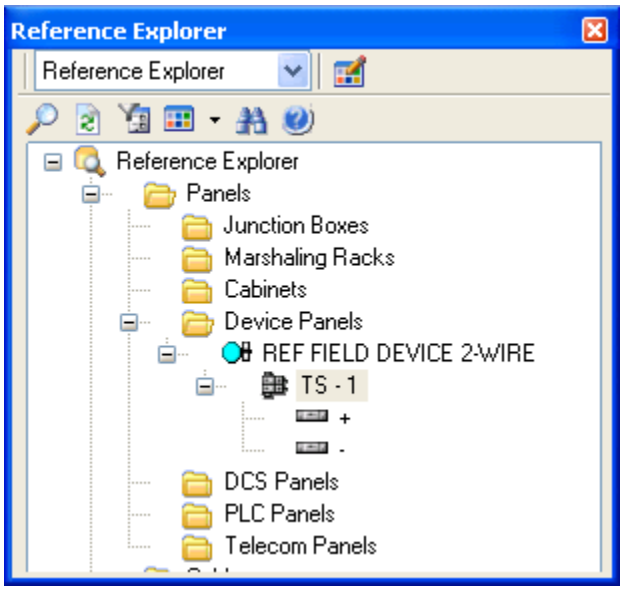
g. Click **Save**.

h. Click **Create**.

i. On the **Terminal Strip Properties** dialog box, in the **Terminal strip** box, type **TS - 1**.

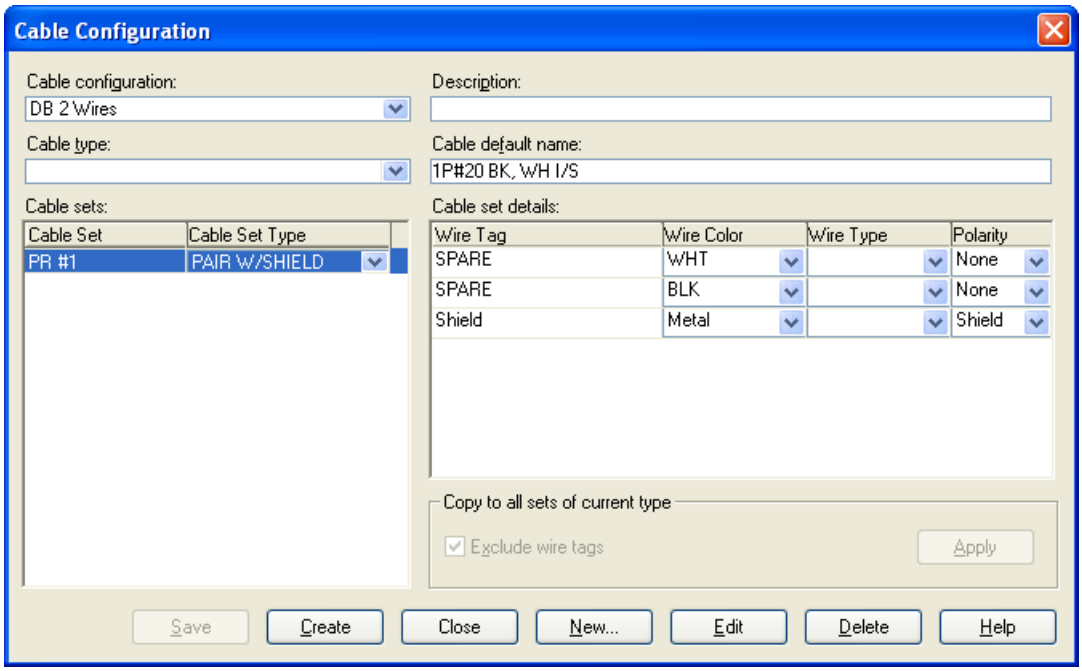
j. Click **OK** and double-click the panel and then the strip to expand the panel-strip-terminal hierarchy in the **Reference Explorer**.

The window should appear as shown:



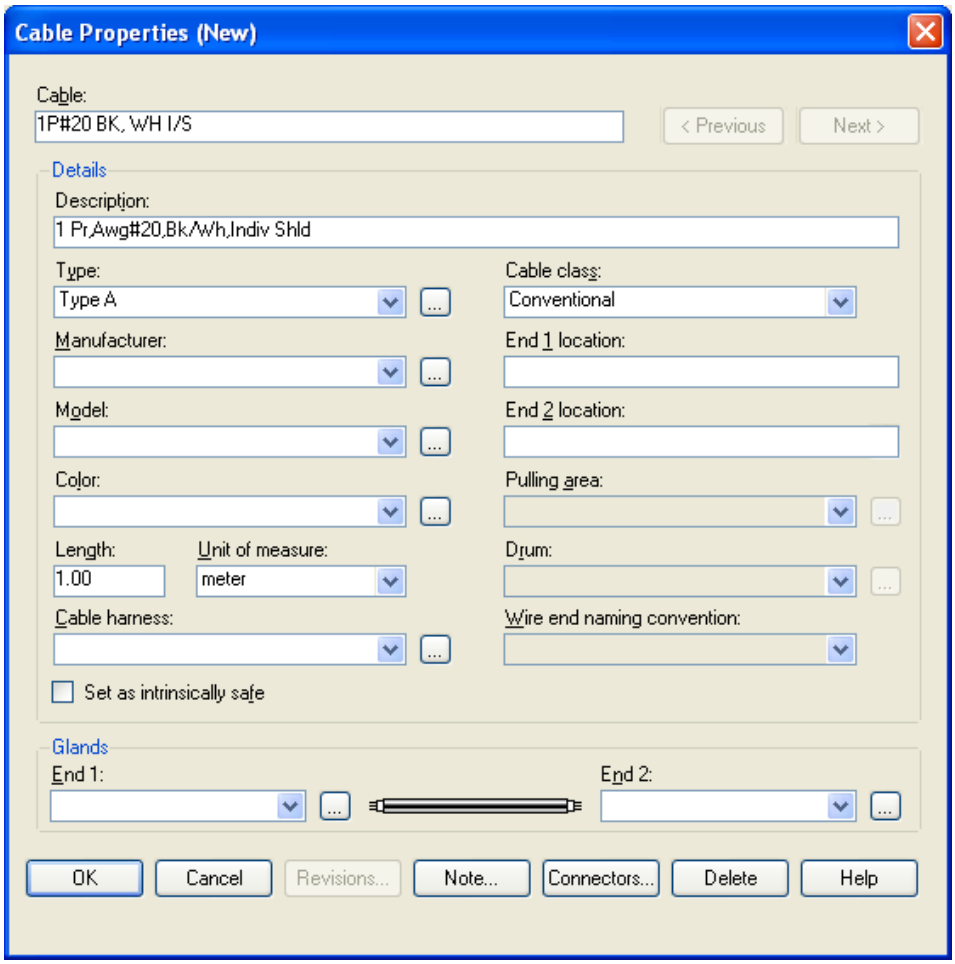
8. Create a reference cable as follows:
 a. Right-click the **Cables** folder and on the shortcut menu, click **New** > **Cable**.
 b. On the **Cable Configuration** dialog box, click **New**.
 c. On the **New Cable Configuration** dialog box, enter **1** as the total number of sets and select **PAIR W/SHIELD** as the default cable set type.
 d. Click **OK**.
 e. In the **Cable configuration** box, type **DP 2 Wires**.
 f. In the **Cable Set** column, type **PR #1**.
 g. In the **Cable default name** box, type **1P#20 BK,WH I/S**.
 h. Under **Cable set wire details**, make sure that the wire tag label is **SPARE** for the first two wire tags and **Shield** for the third wire tag.

 The dialog box should now appear as shown:

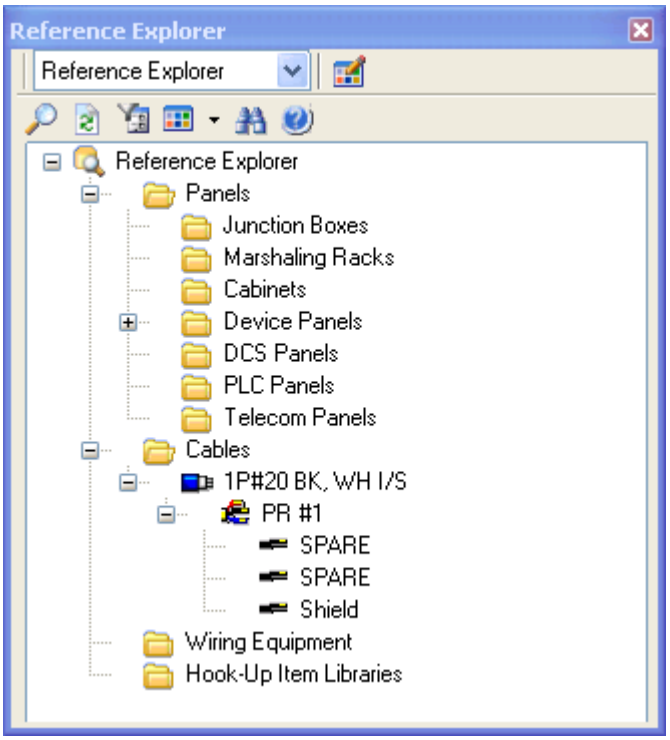


 i. Click **Save**.
 j. Click **Create**.

k. On the **Cable Properties** dialog box, enter information as shown and click **OK**.



9. In the **Reference Explorer**, display your cable, which should appear as shown.

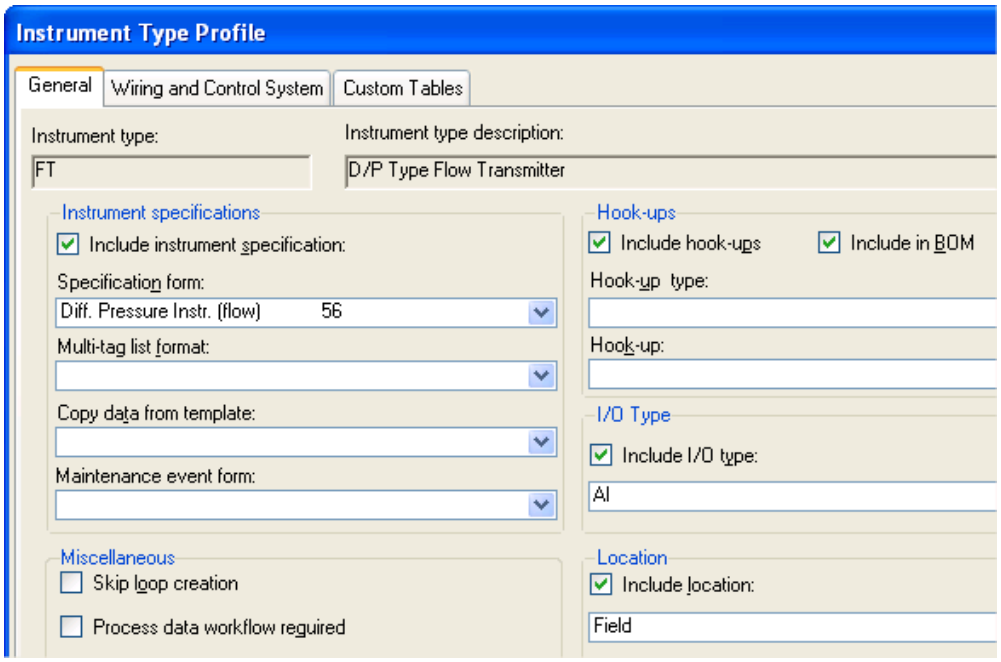


Define the FT (D/P Type Flow Transmitter) Instrument Type Profile

After completing this procedure, when you create a flow transmitter later, the software will apply the instrument type profile settings and automatically create a two-terminal field device with two wires connected to it.

1. Make sure that the Instrument Index module is open.
2. Click **Tables** > **Instrument Types**.
3. From the **Process function** list, select **Flow**.
4. Scroll down the **Instrument Type** list and select instrument type **FT** (with description **D/P Type Flow Transmitter**) which you defined in a previous task.
5. Click **Profile** to open the **Instrument Type Profile** dialog box.
6. On the **General** tab, do the following:
 a. In the **Instrument specifications** group box, select **Include instrument specification**.
 b. From the **Specification form** list, select **Diff. Pressure Instr. (flow) 56**.
 c. In the **Hook-ups** group box, select **Include hook-ups** and **Include in BOM**.
 d. In the **I/O type** group box, select **Include I/O type** and from the list, select **AI**.
 e. In the **Location** group box, select **Include location** and from the list, select **Field**.

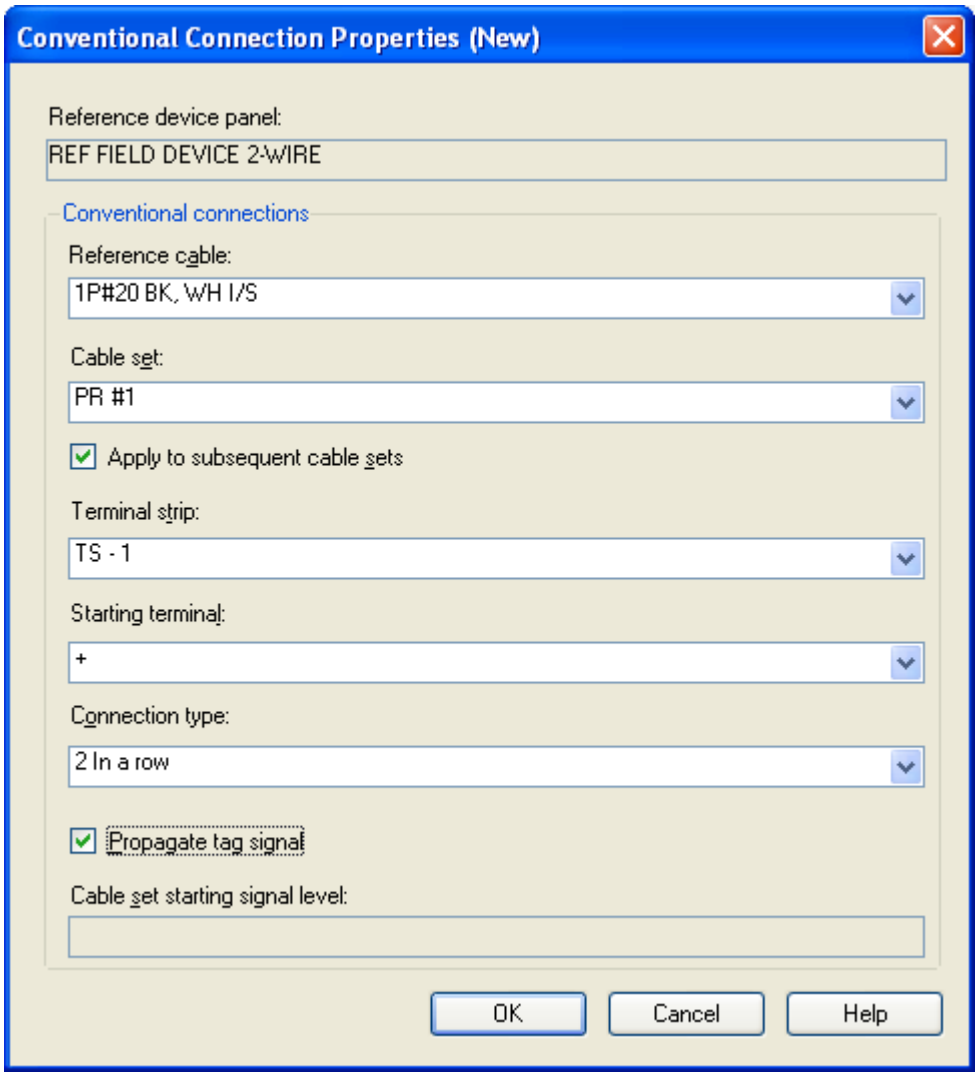
The **General** tab of the **Instrument Type Profile** dialog box should now appear as shown:



7. Click **Apply**.
8. Click the **Wiring and Control System** tab.

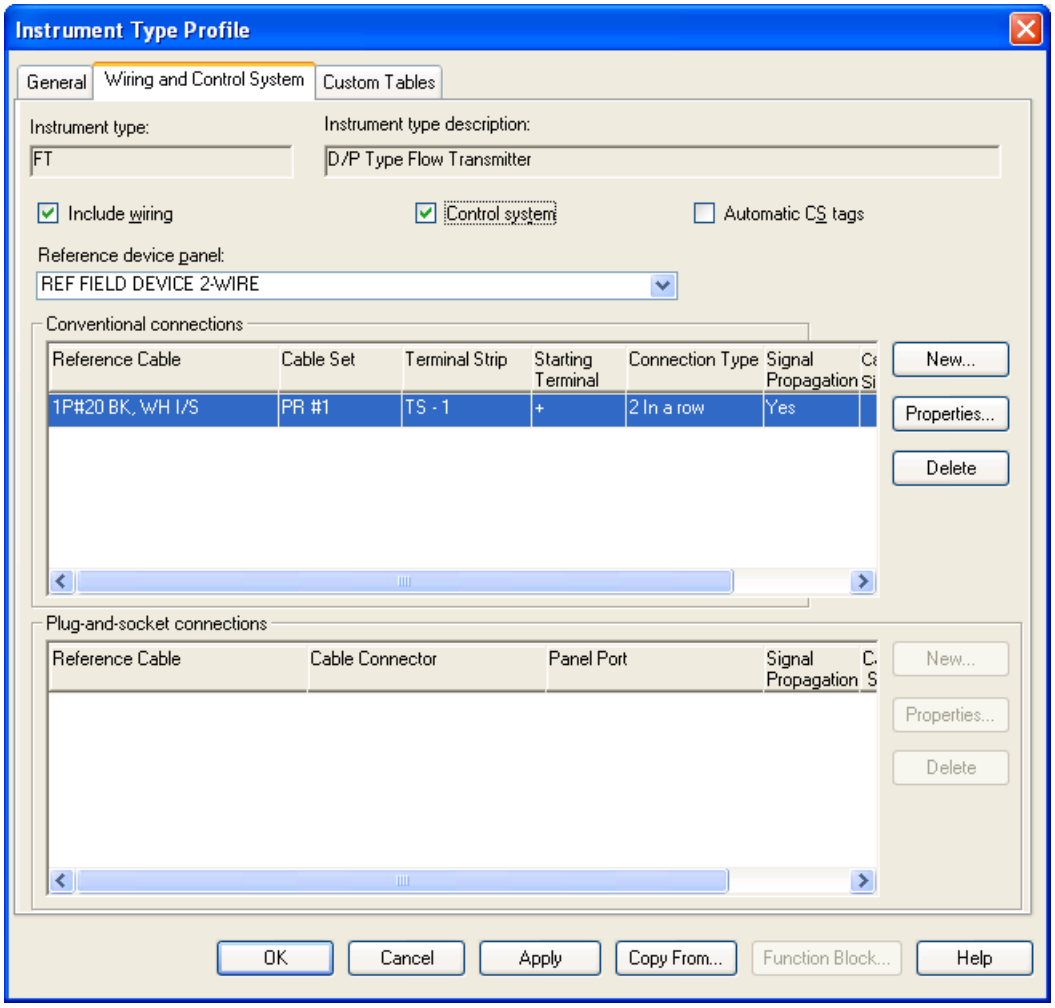
 TIP The parameters you will enter on this tab are based on the assumption that the device panel to be created will be a two-terminal field device with two wires connected to it.
9. Do the following to define basic parameters and select a reference device panel for tags that you will base on this instrument type:
 a. Select **Include wiring** and **Control system**.
 b. From the **Reference device panel** list, select **REF FIELD DEVICE 2-WIRE**.
10. Do the following to select the reference device cable and its connections:
 a. In the **Conventional connections** group box, click **New**.
 b. On the **Conventional Connection Properties** dialog box, from the **Reference cable** list, select **1P#20 BK, WH I/S**.
 c. From the **Cable set** list, select **PR #1**.
 d. Accept the default selection of **Apply to subsequent cable sets**.
 e. From the **Terminal strip** list, select **TS - 1**.
 f. From the **Starting terminal** list, select +.
 g. From the **Connection type** list, select **2 In a row**.
 h. Accept the default selection of **Propagate tag signal**.

The **Conventional Connection Properties** dialog box should now appear as shown:



i. Click **OK** to save your values and close the **Conventional Connection Properties** dialog box.

The **Wiring and Control System** tab of the **Instrument Type Profile** dialog box should now appear as shown:

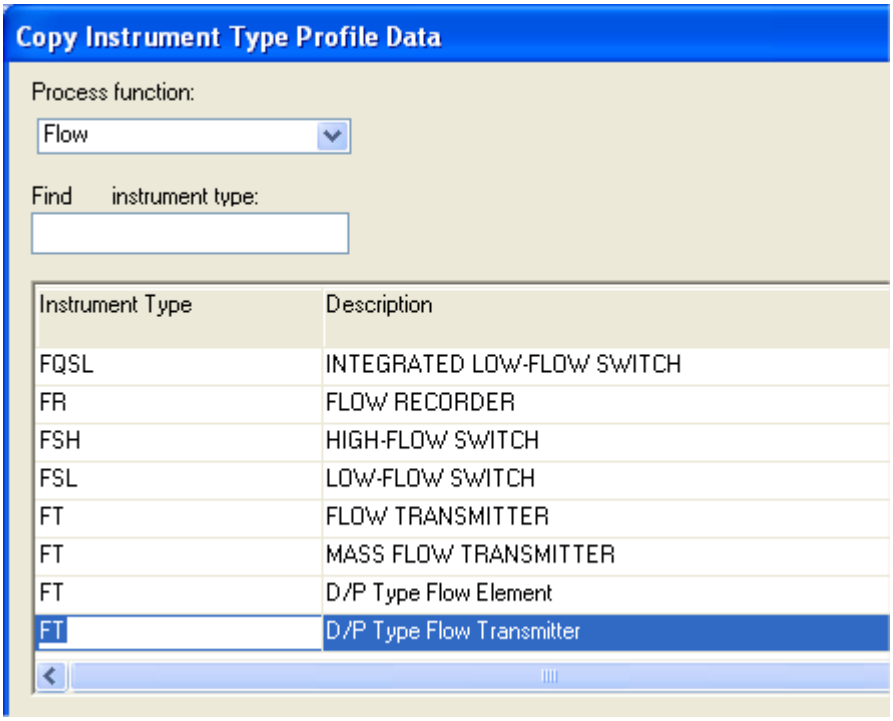


11. Click **OK** to return to the **Instrument Types** dialog box.
12. Click **Apply**.

Define the FY and TY (I/P Transducer) Instrument Type Profiles

1. On the **Instrument Types** dialog box, from the **Process function** list, select **General**.
2. Scroll down the **Instrument Type** list and select instrument type **FY** (with description **I/P Transducer**).
3. Click **Profile** to open the **Instrument Type Profile** dialog box.
4. On the **General** tab, click **Copy From** and then, do the following:
 a. On the **Copy Instrument Type Profile Data** dialog box, from the **Process function** list, select **Flow**.

b. Select **FT D/P Type Flow Transmitter**, for which you defined a profile in the previous procedure.

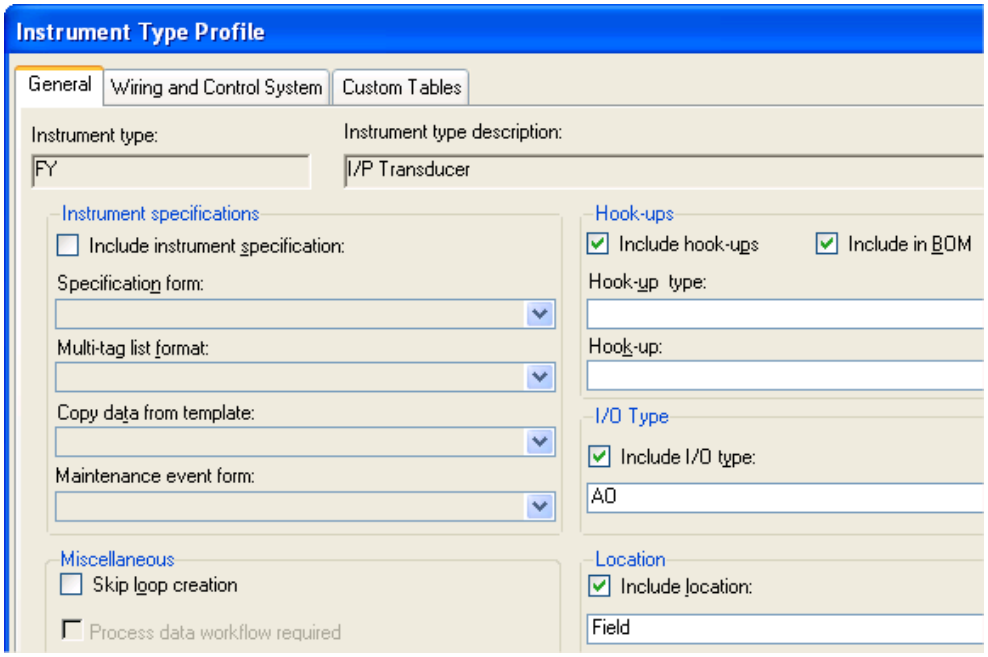


c. Click **OK**.

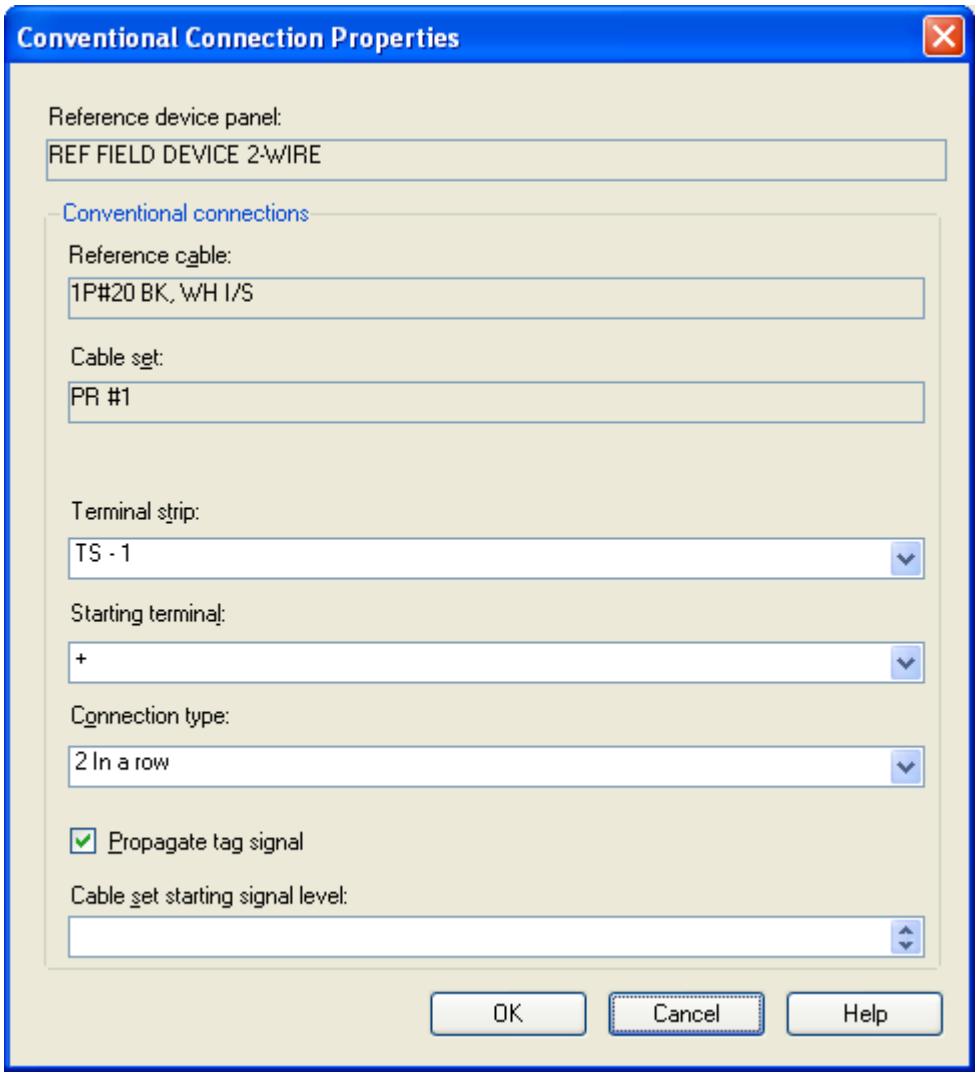
d. In the message box notifying you that you cannot copy specification data, click **OK**.

5. On the **General** tab, in the **I/O type** group box, select **AO** from the list.

The **General** tab of the **Instrument Type Profile** dialog box should now appear as shown:



6. Click the **Wiring and Control System** tab, and beside the **Conventional connections** group box options, click **Properties** and ensure that the wiring profile settings have been copied as shown.

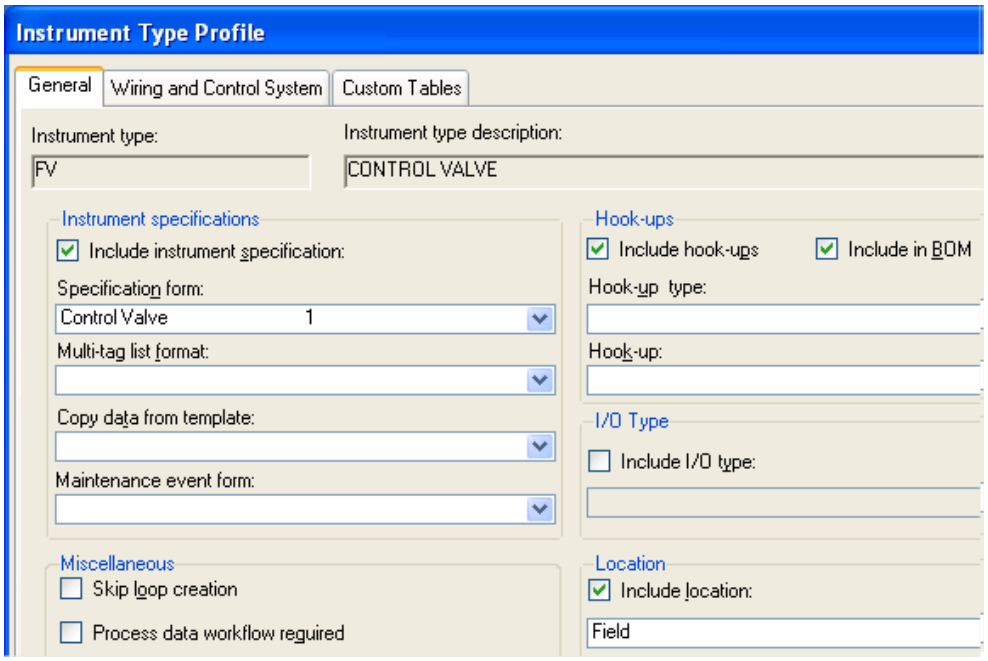


7. Without making any changes, click **OK** to close the **Conventional Connection Properties** dialog box.
8. On the **Instrument Type Profile** dialog box, click **OK** to return to the **Instrument Types** dialog box.
9. Click **Apply**.
10. Select instrument type **TY** (with description **I/P Transducer**) and copy the profile definitions from **FY**.

Define the FV and LV (CONTROL VALVE) Instrument Type Profiles

1. On the **Instrument Types** dialog box, from the **Process function** list, select **Control Valve**.
2. Scroll down the **Instrument Type** list and select instrument type **FV** (with description **CONTROL VALVE**).
3. Click **Profile** to open the **Instrument Type Profile** dialog box.
4. On the **General** tab, do the following:
 a. In the **Instrument specifications** group box, select **Include instrument specification**.
 b. From the **Specification form** list, select **Control Valve 1**.
 c. In the **Hook-ups** group box, select **Include hook-ups** and **Include in BOM**.
 d. In the **Location** group box, select **Include location** and from the list, select **Field**.

 The **General** tab of the **Instrument Type Profile** dialog box should now appear as shown:



5. Click **OK** to return to the **Instrument Types** dialog box.
6. Click **Apply**.
7. Select instrument type **LV** (with description **CONTROL VALVE**) and copy the profile definitions from **FV**.

Define Additional Instrument Type Profiles

Carefully read the notes below the tables before beginning.

Create the following instrument types:

Process Function	Instrument Type	Description	Spec	Hook-Ups	I/O Type	Control System	Wiring
General	LY	I/P Transducer	–	Y	AO	Y	Y
General	HY	I/P Transducer	–	–	AO	Y	Y
Flow	FI	Local Flow Indicator	–	–	–	–	Y
Control Valve	HV	Control Valve	–	–	–	–	–

Modify the following instrument types:

Process Function	Instrument Type	Description	Spec	Hook-Ups	I/O Type	Control System	Wiring
Flow	FE	D/P Type Flow Element	–	Y	–	–	–
Level	LT	Level Transmitter	–	Y	AI	Y	Y
Pressure	PI	Pressure Gauge	–	Y	–	–	–
Pressure	PSH	High-Pressure Switch	24	Y	DI	Y	Y
Pressure	PT	Pressure Transmitter	–	–	AI	Y	Y
Temperature	TE	Thermocouple	–	Y	–	–	Y
Temperature	TI	Bi-Metal Thermometer	–	Y	–	–	–
Temperature	TW	Thermowell	–	–	–	–	–
Control Valve	TV	Control Valve	1	Y	–	–	–

NOTES

- The **Spec** column indicates the form number you need to select for certain instruments on the **General** tab of the **Instrument Type Profile** dialog box.
- Under **Hook-Ups**, if the value in the table is **Y**, select the **Include hook-up**s and **Include in BOM** check boxes on the **General** tab of the **Instrument Type Profile** dialog box.
- Under **Wiring**, if the value in the table is **Y**, use the wiring definitions that you applied for **FT** (**D/P Type Flow Transmitter**).
- Under **Control System**, if the value in the table is **Y**, select the **Control system** check box on the **Wiring and Control System** tab of the **Instrument Type Profile** dialog box.

- For all instrument types in the table, select **Field** as the value of the **Location** property**.**

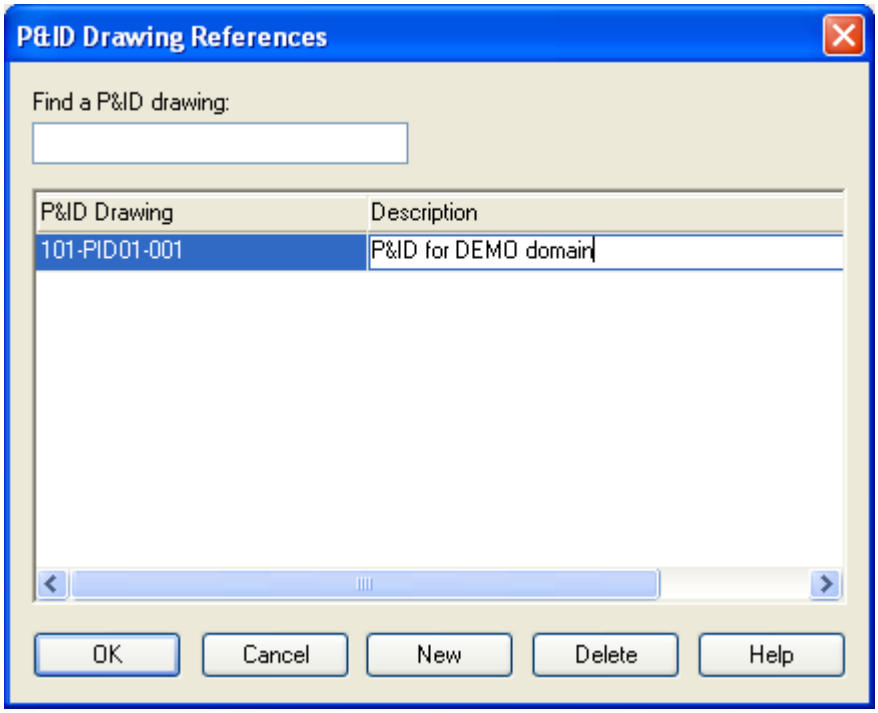
Task 2 — Using Supporting Tables to Add Values to Select Lists

This task enables you to categorize various instrument properties in select lists. These select lists appear in the database in supporting tables. In the SmartPlant Instrumentation interface, supporting tables are represented by dialog boxes that allow you to create, delete, and edit various properties that apply to SmartPlant Instrumentation loops or instrument tags.

Create a P&ID Drawing Document Number

1. Click **Tables** > **P&ID Drawing References**.
2. On the **P&ID Drawing References** dialog box, click **New** to add a new data row.
3. Complete the information for the P&ID document drawing number and description as follows:
 a. In the **P&ID Drawing** column, type **101-PID01-001**.
 b. In the **Description** column, type **P&ID for DEMO domain**.

 The dialog box should appear as shown:



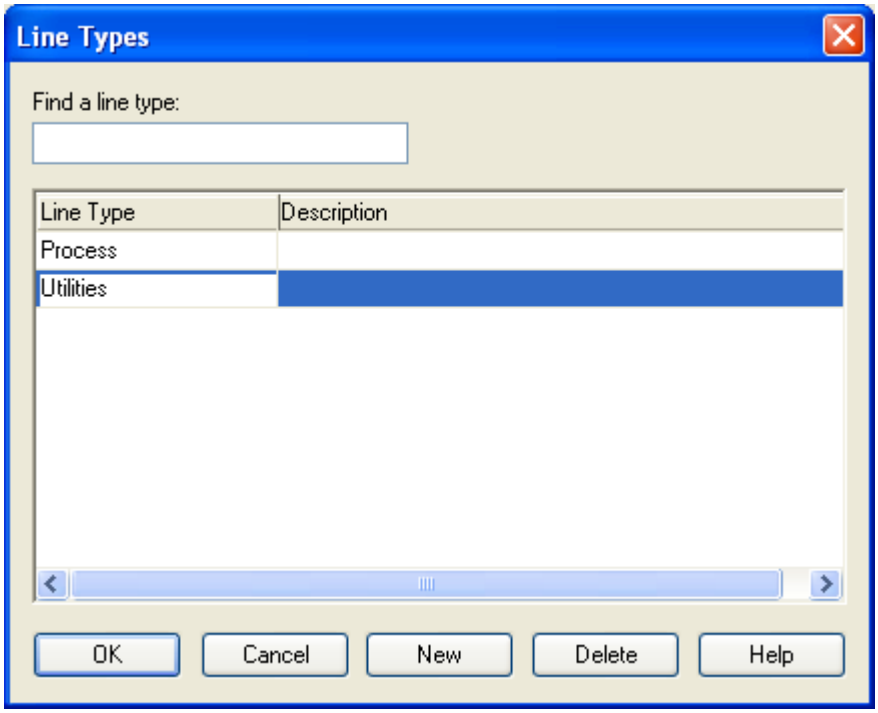
4. Click **OK** to save the data and close the dialog box.

Create Lines (Piping)

Lines and line data are defined in the Process Data module since these definitions are usually the responsibility of process engineers. After you define line numbers, it is possible to associate instruments with the lines on which they are located.

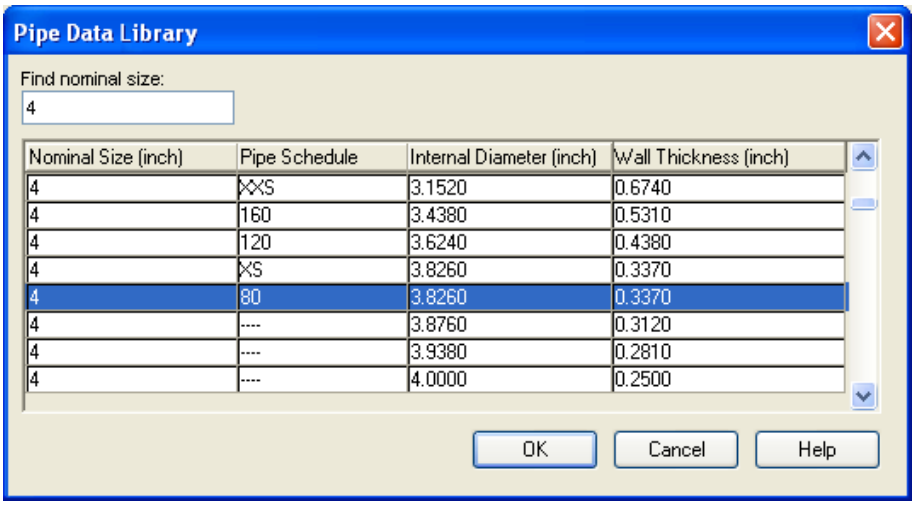
1. Click **Tables** > **Lines**.
2. On the **Lines** dialog box, beside the **Line type** list arrow, click ... to open the **Line Types** dialog box and do the following:
 a. Click **New** to add a new data row and under **Line Type**, type **Process.**
 b. Click **New** to add another data row and under **Line Type**, type **Utilities**.

 The dialog box should appear as shown:



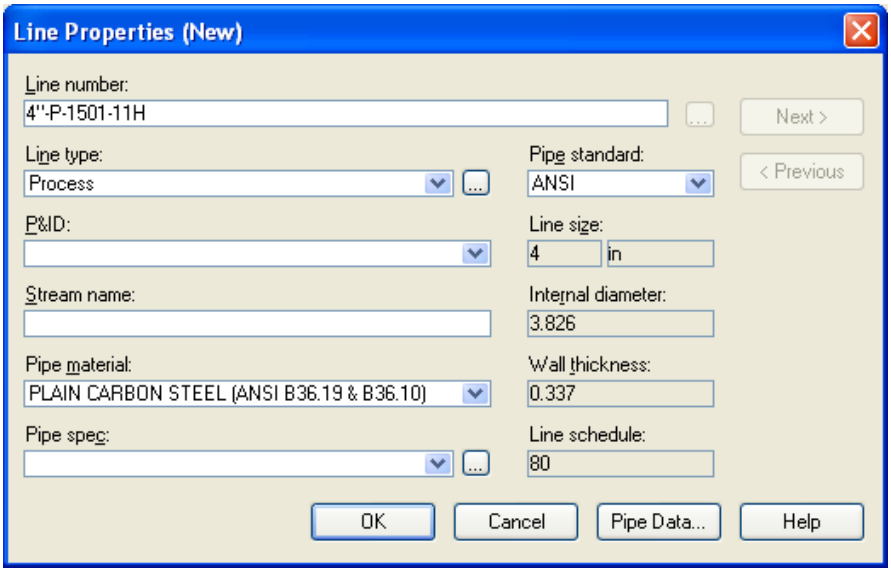
3. Click **OK** to save the data and close the dialog box.
4. On the **Lines** dialog box, from **Line type** list, select **Process**.
5. Click **New** to open the **Line Properties (New)** dialog box.
6. Add the following information:
 a. In the **Line number** box, type **4"-P-1501-11H**.
 b. From the **Pipe material** list, select **PLAIN CARBON STEEL (ANSI…)**.
 c. From the **Pipe standard** list, select **ANSI**.
7. On the **Pipe Data Library** dialog box, do the following:
 a. In the **Find nominal size** box, type **4** so that you can easily select **4 inches** as the nominal size.

b. Select the pertinent pipe data as shown.



c. Click **OK** to close the **Pipe Data Library** dialog box and return to the **Line Properties** dialog box.

All other pipe details are entered automatically and the **Line Properties** dialog box should appear as shown:



8. Click **OK** to select the line settings and to return to the **Lines** dialog box.
9. Add another line **4"-P-1502-11H** with the same settings.

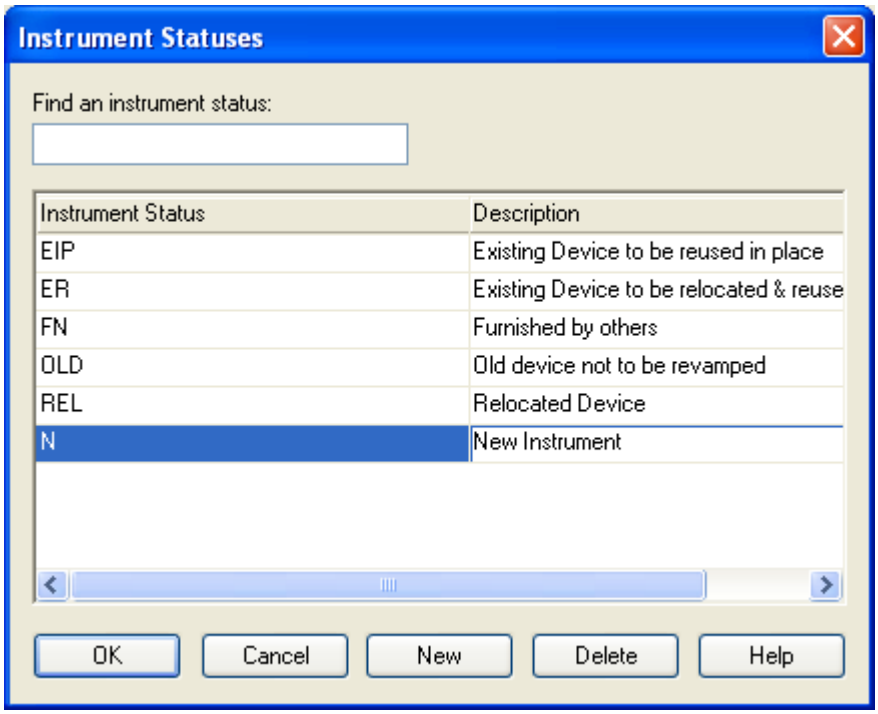
NOTE You can also create lines in the Process Data module.

The remaining objectives in this task deal with entering additional data in the supporting tables. These include **Status**, **I/O Type**, **Location**, **Manufacturer**, and **Model**.

Enter Instrument Status Data

1. Click **Tables** > **Instrument Statuses**.
2. On the **Instrument Statuses** dialog box, do the following:
 a. Click **New** to add a new data row.
 b. In the **Instrument Status** column, type **N**.
 c. In the **Description** column, type **New Instrument**.

 The dialog box should appear as shown:



3. Click **OK**.

Enter I/O Type Data

1. Click **Tables** > **I/O Types**.
2. On the **I/O Types** dialog box, do the following:
 a. Click **New** to add a new data row.
 b. In the **I/O Type** column, type **T/C**.
 c. In the **Description** column, type **T/C Input**.
3. Click **OK**.

Enter Location Data

1. Click **Tables** > **Instrument Locations**.
2. On the **Instrument Locations** dialog box, do the following:
 a. Select the **Field** instrument location.
 b. In the **Description** column, type **Installed in the field**.
3. Click **OK**.

Enter Manufacturer Data

1. Click **Tables** > **Instrument Manufacturers**.
2. On the **Instrument Manufacturers** dialog box, do the following:
 a. Click **New** to add a new row.
 b. In the **Instrument Manufacturer** column, type **Rosemount**.
 c. Enter additional manufacturers **ASCO** and **Ashcroft**.
3. Click **OK**.

Enter Model Data

1. Click **Tables** > **Instrument Models**.
2. On the **Instrument Models** dialog box, do the following:
 a. From the **Manufacturer** list, select **ASCO**.
 b. Click **New** to add a new data row.
 c. In the **Instrument Model** column, type **8142B12**.
 d. In the **Process Function** column, select **General** from the list.
 e. If desired, type a model description.
3. Enter additional models for various manufacturers as shown:

Manufacturer	Model	Process Function
ASCO	8351B23	General
ASHCROFT	EVERY-ANGLE-12/01	General
	EVERY-ANGLE-13/02	General
	MGS - 136	General
ROSEMOUNT	1151DP4E22S2B1M2	General
	3051S1256	General

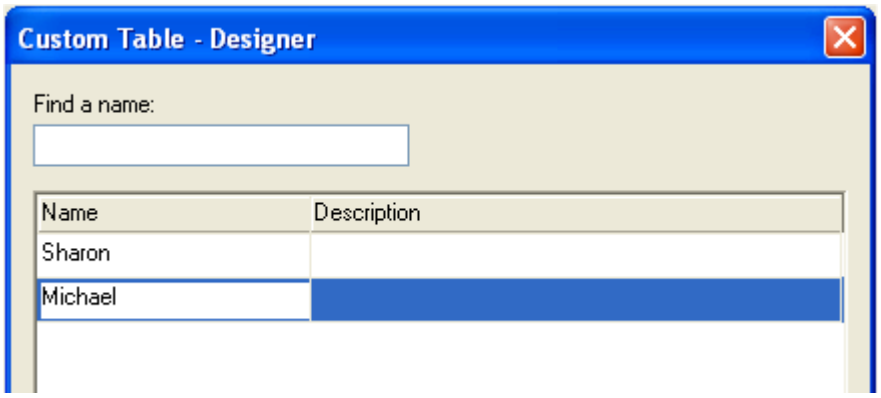
4. Click **OK**.

Enter Values in the Custom Tables

You are now going to enter values in the custom tables that you defined in the Administration module.

1. Click **Tables** > **Custom Tables** > **Designer**.
2. On the **Custom Table - Designer** dialog box, do the following:
 a. Click **New** to add a new data row.
 b. In the **Name** column, type a designer name.
 c. Add another designer.

 The dialog box should appear similar to the example shown (the values that you enter can be different):

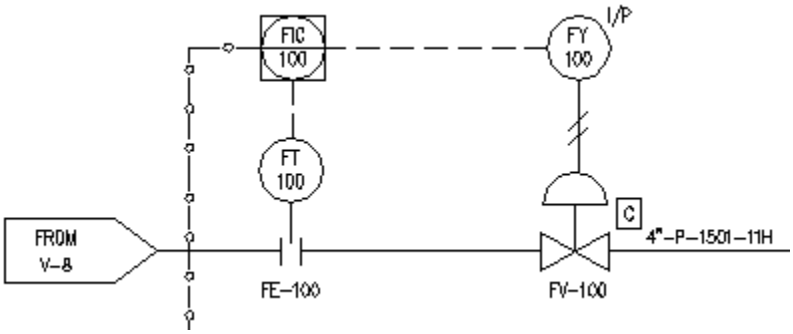


3. Click **OK**.
4. Repeat the above steps for the **Technician** and **Cell Number** custom tables.

This completes the basic supporting table data required for this tutorial. If you want to add more data, do so according to the above instructions.

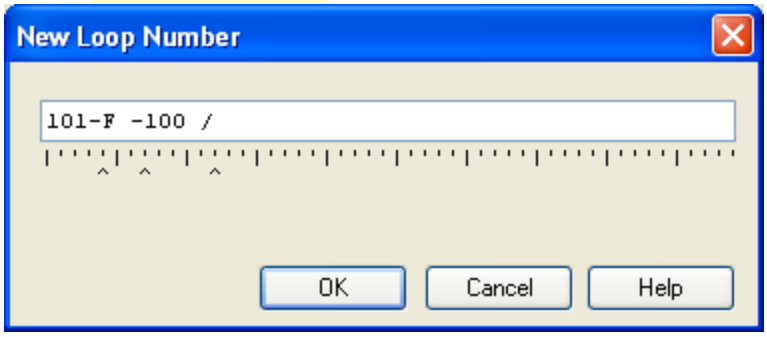
Task 3 — Creating Loops with Tags Using the Domain Explorer

In this task, you are going to use the **Domain Explorer** options to create flow control loop **101-F-100** with its tags as shown on the P&ID.



Define a Loop and Add Tag Numbers

1. Click **Tools** > **Domain Explorer**.
2. Under **Domain Explorer**, expand the plant hierarchy as follows: **New Refinery** > **Crude Area** > **Crude Unit 1**.
3. Under **Crude Unit 1**, right-click the **Loops** folder, and on the shortcut menu, click **New** > **Loop**.
4. On the **New Loop Number** dialog box, type the loop number as shown.

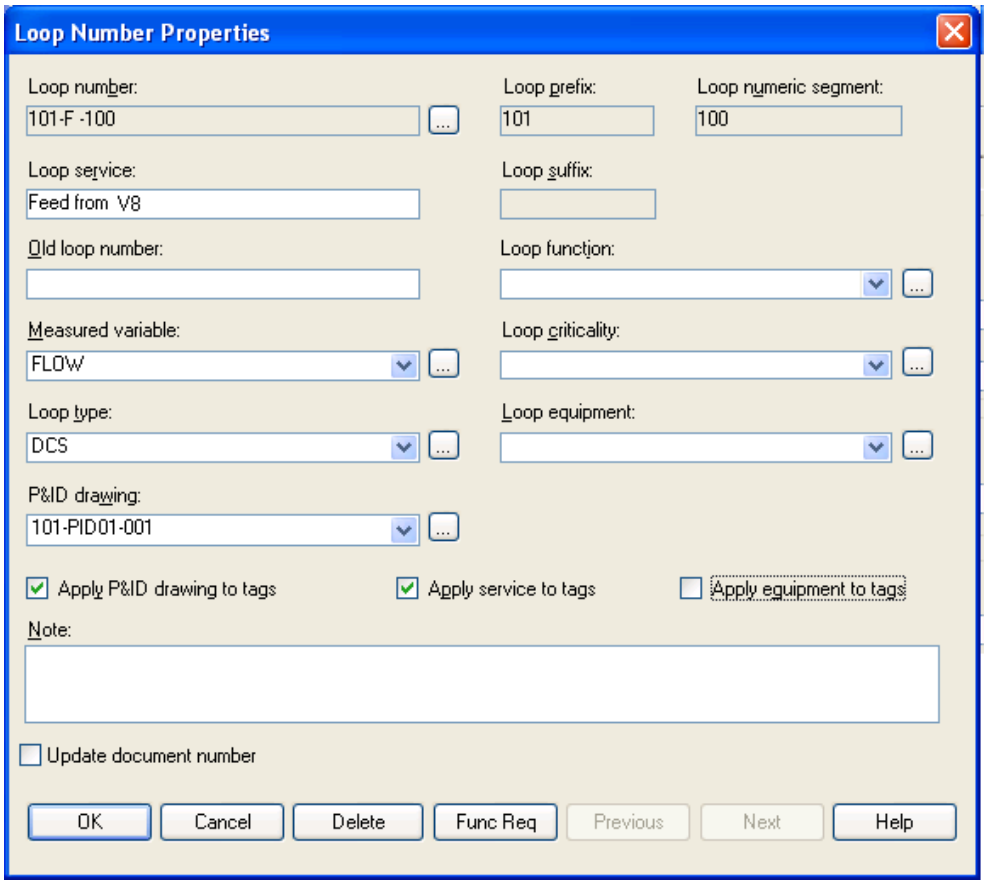


TIPS

- Use the **Tab** key to move the cursor to the character separators.
- The first segment of the loop number is derived from the unit number as you defined it in the loop naming convention and cannot be edited.

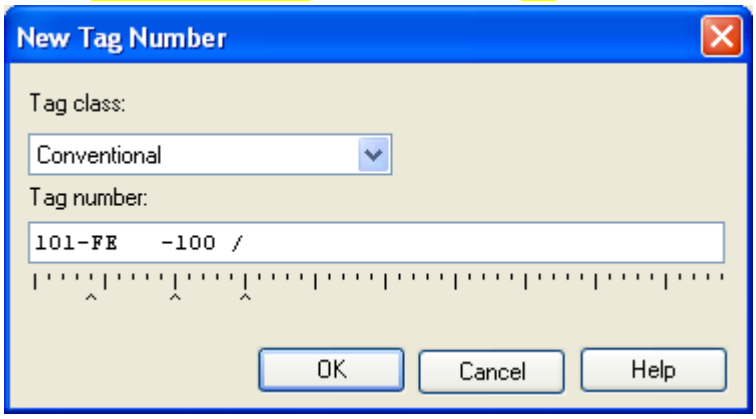
5. Click **OK**.
6. On the **Loop Number Properties** dialog box, enter the following information:
 a. In the **Loop service** box, type **Feed from V8**.
 b. From the **Loop type** list, select **DCS**.
 c. From the **P&ID drawing** list, select **101-PID01-001**.
 d. Clear the **Apply equipment to tags** check box.

The dialog box should appear as shown:



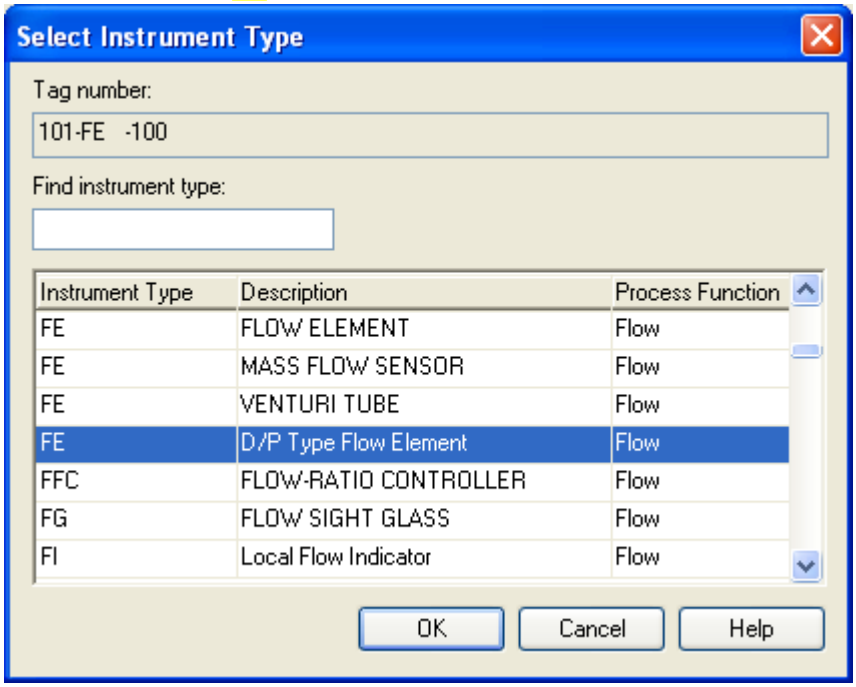
TIP To add new values to supporting tables so that they can become available in the lists, click ... next to the lists.

7. Click **OK**.
8. Click **Yes** when prompted to create a tag number.
9. On the **New Tag Number** dialog box, enter **FE** as the first new tag number.



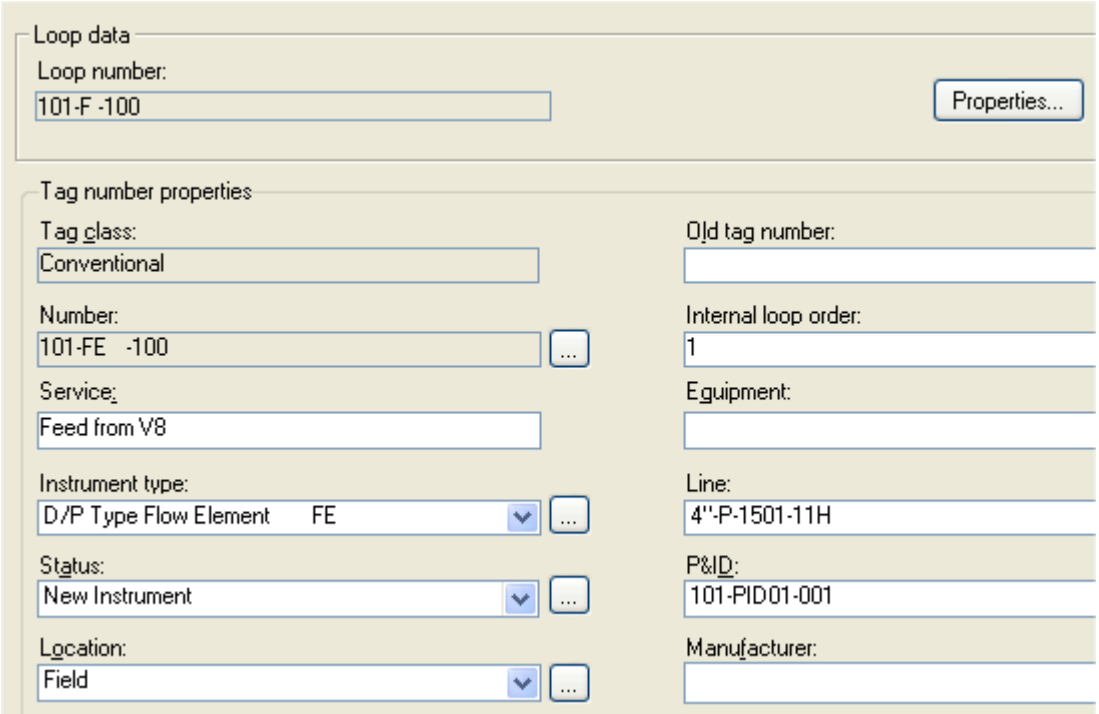
10. Click **OK**.

11. On the **Select Instrument Type** dialog box, select the **D/P Type Flow Element** description for instrument type **FE** as shown.



TIP You need to select an instrument type because more than one instrument type designated by the **FE** acronym exists.

12. Click **OK** to open the **Tag Number Properties** dialog box.
13. Enter tag number data by selecting values from the lists as shown.

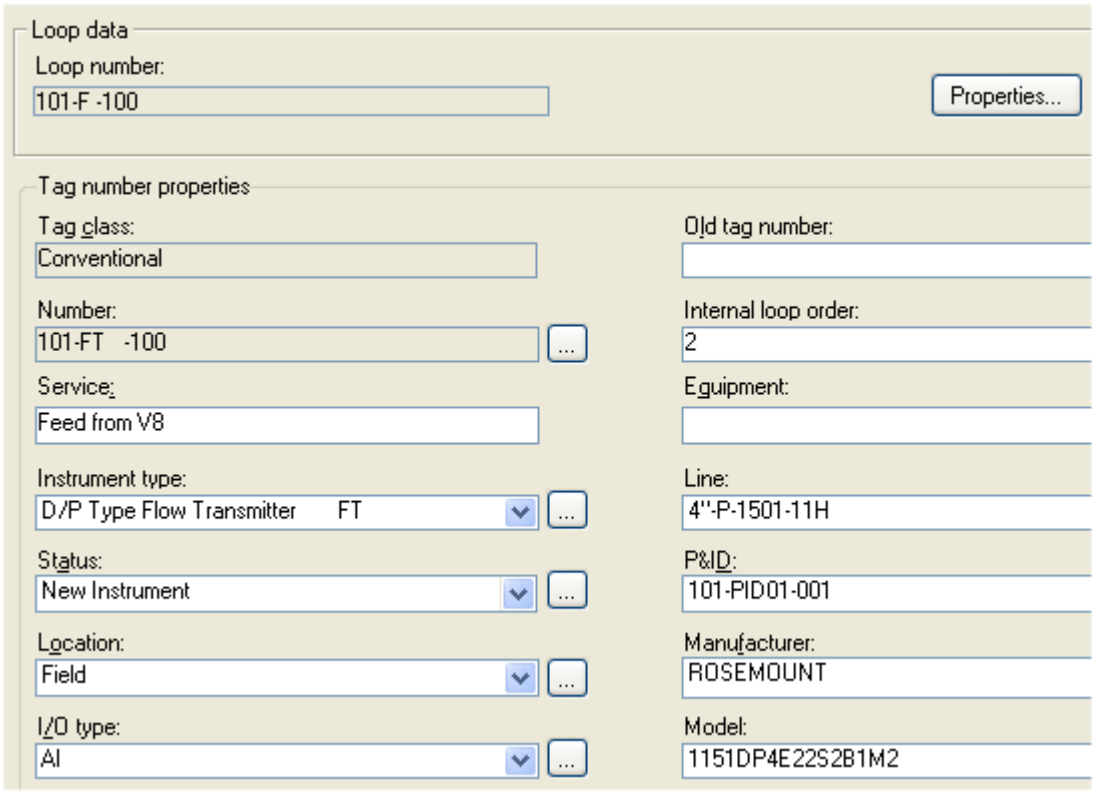


TIP The ellipsis [...] buttons enable you to enter additional data in the supporting data tables. These values then become available in the select lists.

14. Click **Apply** when done.
15. Click **New** to add another tag number to the loop as follows:
 a. On the **New Tag Number** dialog box, enter **FT** so that the tag number is **101-FT -100** and click **OK**.
 b. On the **Select Instrument Type** dialog box, select the **D/P Type Flow Transmitter** description for instrument type **FT**.

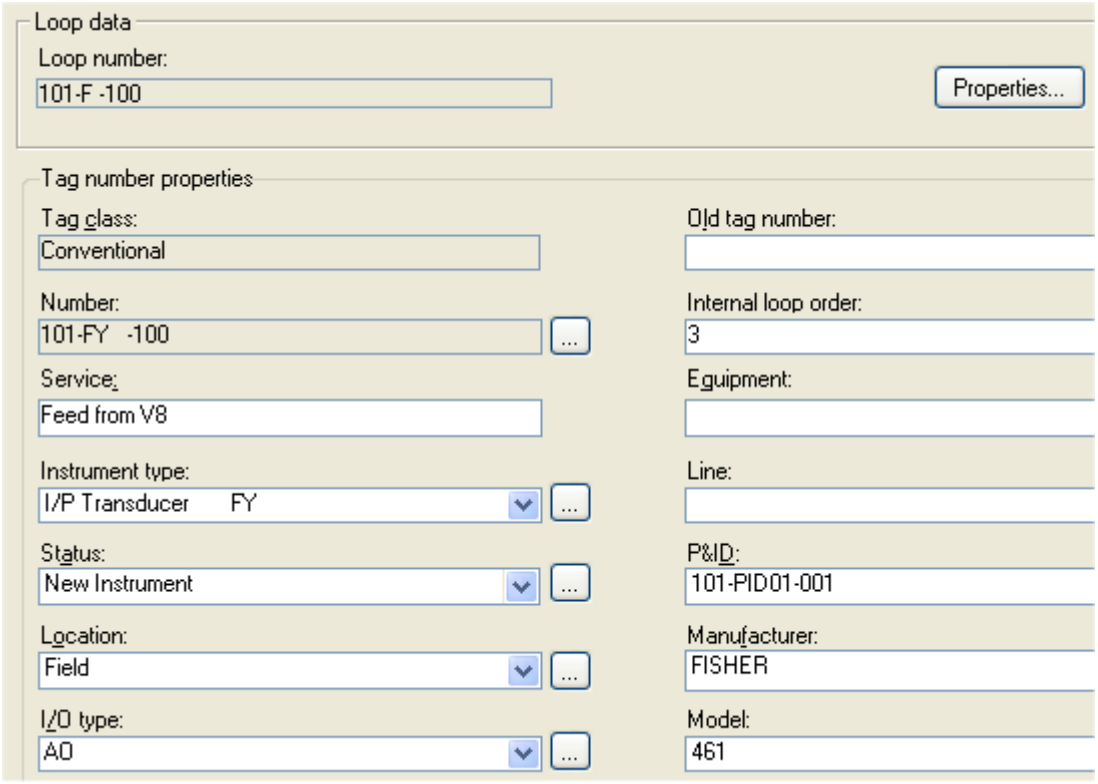
TIP While the software is creating this tag number, the status bar indicates automatic device panel creation.

 c. On the **Tag Number Properties** dialog box, complete the definitions as shown.



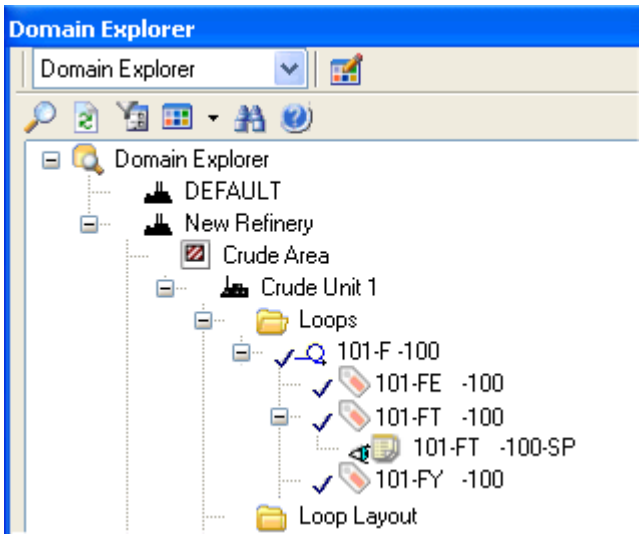
16. Create tag number **101-FY -100** as follows:
 a. Select the **I/P Transducer** description for instrument type **FY** (**General** process function).

b. On the **Tag Number Properties** dialog box, complete the definitions as shown.



17. After completing all your tag definitions, click **OK** to close the dialog box.

In the **Domain Explorer**, your loop and instruments should now appear as shown:

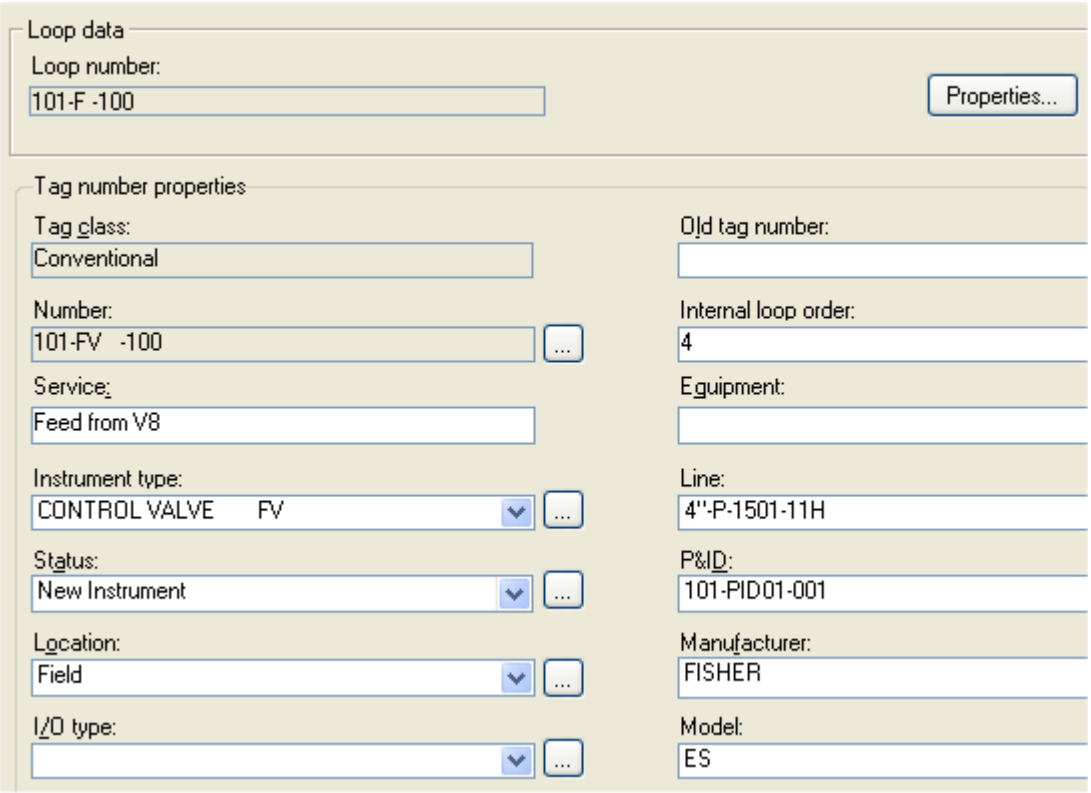


NOTE The spec document number **101-FT -100-SP** has been created automatically with tag number **101-FT -100** because the instrument type profile of this tag includes a specification.

Add a Tag Number to an Existing Loop

In this procedure, you will add a control valve **FV** to loop **101-F -100**.

1. Press **F7** to open the **Domain Explorer**.
2. Expand the plant hierarchy as follows: **New Refinery** > **Crude Area** > **Crude Unit 1**.
3. Under **Crude Unit 1**, right-click the **Instruments** folder, and on the shortcut menu, click **New** > **Instrument**.
4. On the **New Tag Number** dialog box, type tag number **101-FV -100** (using the **Tab** key to ensure that the number of spaces is correct) and click **OK**.
5. On the **Loop Name** dialog box, accept the loop name **101-F -100** and click **OK**.
6. On the **Tag Number Properties** dialog box, define the tag number as shown.

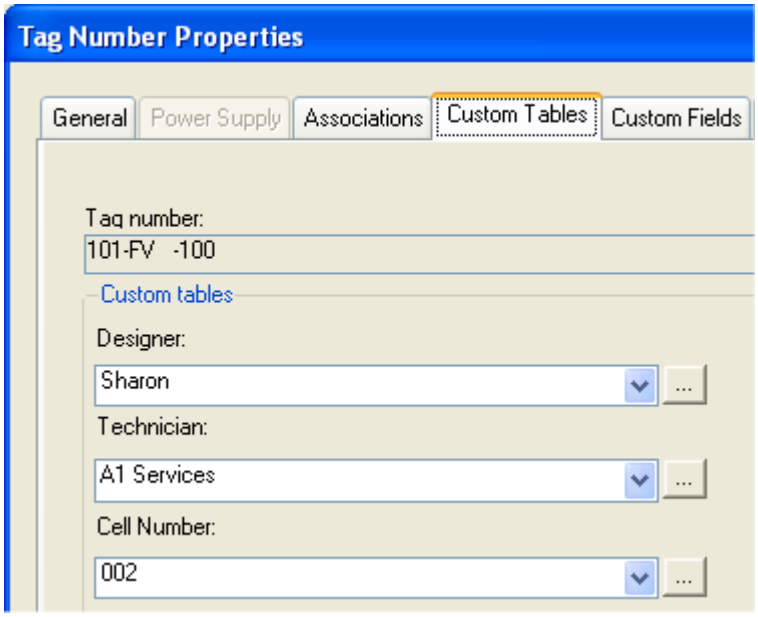


7. Click **OK**.
8. In the **Domain Explorer**, under **Crude Unit 1**, select the **Instruments** folder and on the **Domain Explorer** toolbar, click  to refresh the display of items.

Assign Custom Table and Custom Field Values to a Tag Number

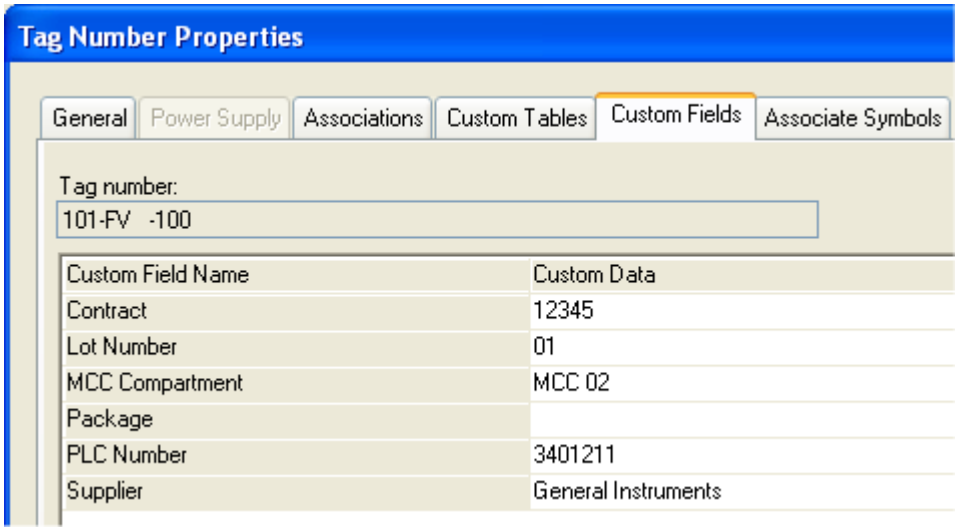
In this procedure, you are going to assign custom table and custom field values to control valve **101-FV -100**.

1. Open the **Tag Number Properties** dialog box for control valve **101-FV -100**.
2. Click the **Custom Tables** tab.
3. From the **Designer** list, select a designer.
4. Select values for the **Technician** and **Cell number** lists as shown in the following example (the values that you enter can be different).



5. Click **Apply**.
6. Click the **Custom Fields** tab.

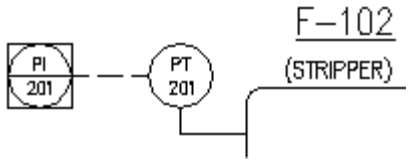
7. Enter information as shown in the following example (the values that you enter can be different).



8. Click **OK**.

Task 4 — Creating a Loop with Its Tag Using the Instrument Index Module

In this task, you are going to use the Instrument Index module to create loop **101-P -201** with its tag as shown.

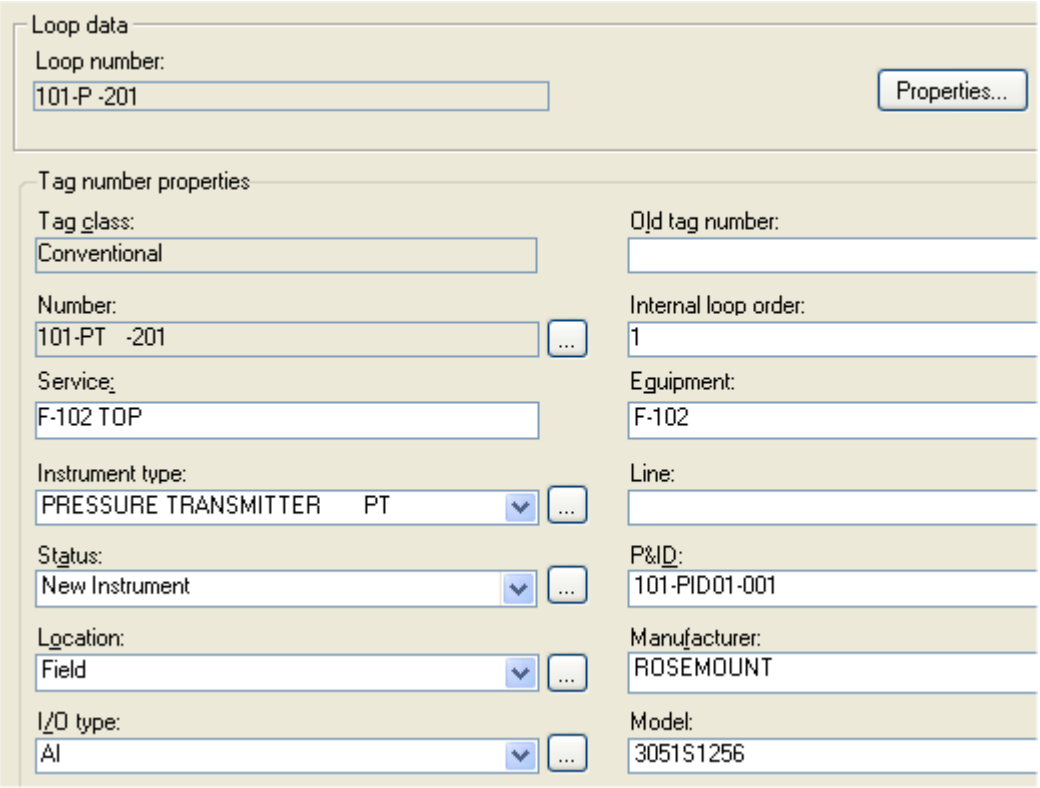


Create a Loop with a Single Instrument Tag

1. Close the **Domain Explorer**.
2. Click **Module** > **Instrument Index**.
3. In the **Instrument Index Module** window, on the toolbar, click **New Tag** .
4. On the **New Tag Number** dialog box, type tag number **101-PT -201**, and click **OK**.
5. On the **Loop Name** dialog box, accept the loop name **101-P -201** and click **OK**.
6. On the **Loop Number Properties** dialog box, click **OK**.
7. On the **Tag Number Properties** dialog box, do the following to create the equipment that you need:
 a. Beside **Equipment**, click ... to open the **Equipment** dialog box.
 b. From the **Equipment Type** list, select **TANK**.
 c. Click **New** to add a new row.

d. In the **Equipment** column, type **F-102** and click **OK**.

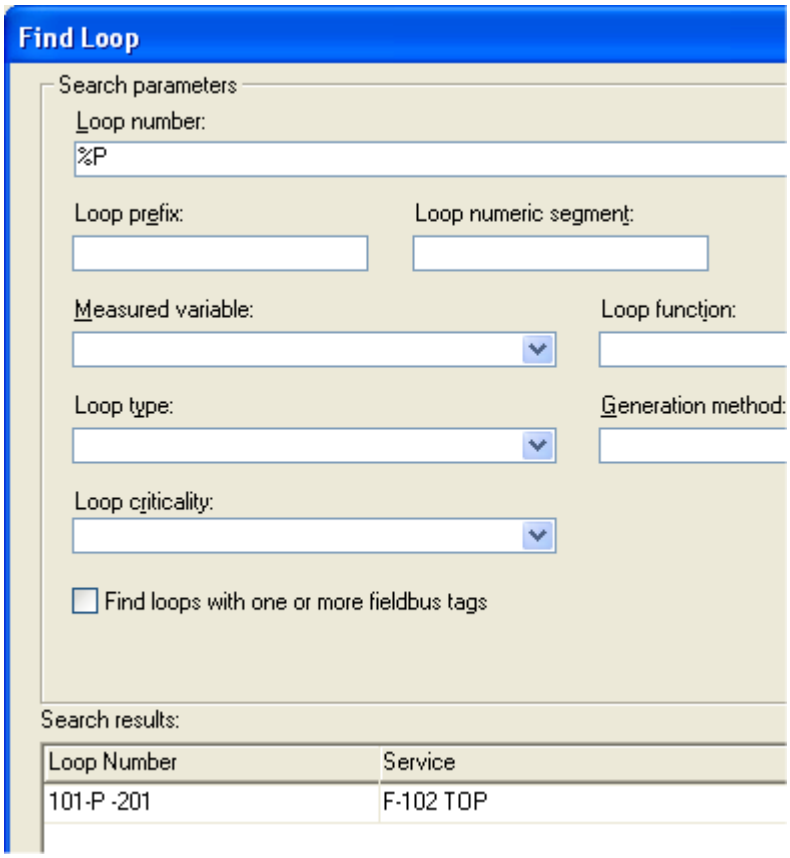
8. Beside **Loop number**, click **Properties** to open the **Loop Number Properties** dialog box.
9. In the **Loop service** box, type **F-102 TOP**.
10. Click **OK**.
11. On the **Tag Number Properties** dialog box, complete the data entry as shown.



12. Click **OK**.
13. In the **Instrument Index Module** window, on the toolbar, click (**Loop Properties**).
14. On the **Enter Loop Number** dialog box, click **Find**.
15. On the **Find Loop** dialog box, in the **Loop number** box, enter **%P**.

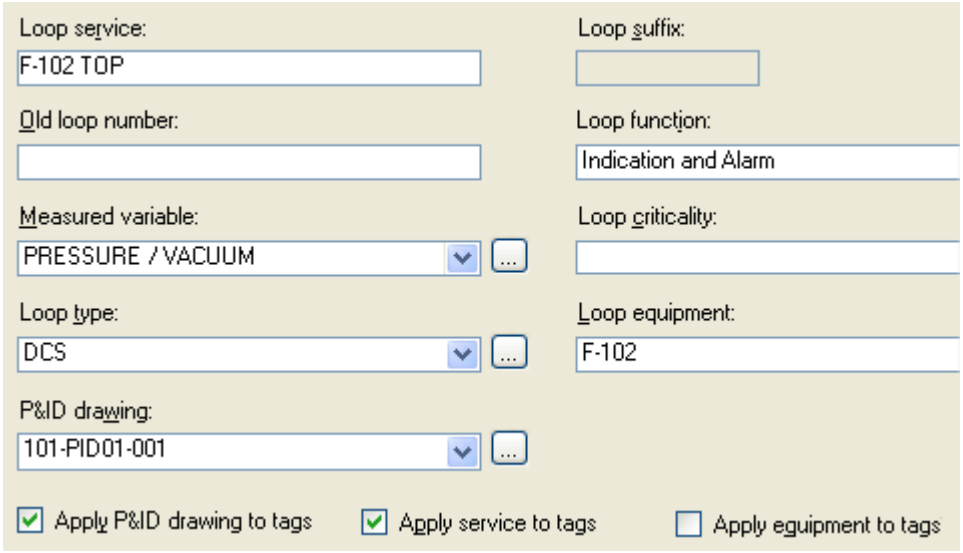
TIP The **%** symbol is a multi-character wildcard which substitutes for any characters that precede or follow the specified character or string.

The **Find Loop** dialog box should display the following result:



16. Select loop **101 P -201** and click **OK**.
17. On the **Loop Number Properties** dialog box, do the following to add a loop function to the **Loop function** list:
 a. Beside **Loop function**, click
 b. On the **Loop Functions** dialog box, click **New** to add a new line.
 c. In the **Loop Function** column, type **IA**.
 d. In the **Description** column, type **Indication and Alarm**.
 e. Click **OK**.
18. Clear the **Apply equipment to tags** check box.

19. Enter the remaining loop data as shown.

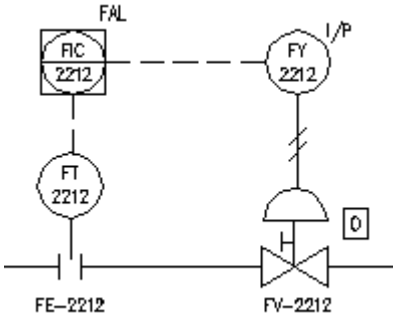


20. Click **OK**, and when prompted to edit tag numbers, click **No**.
21. Press **F7** to open the **Domain Explorer.**
22. Expand **New Refinery** > **Crude Area** > **Crude Unit 1** to displays item folders in **Crude Unit 1**.
23. Expand the **Loops** folder and view loop **101 P -201** and its tag.

Task 5 — Duplicating a Loop with Its Tag Numbers

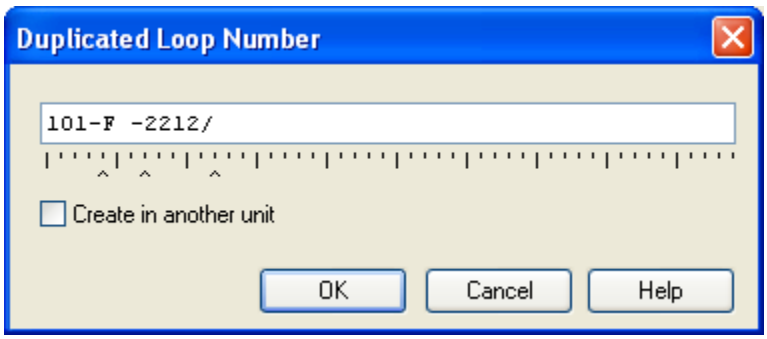
If you want to create a series of control loops that perform similar functions, rather than creating each one from scratch, you can duplicate an existing loop together with all or some of its tag numbers.

In this task, you will duplicate loop **101-F 100** to create loop **101-F -2212** as shown.

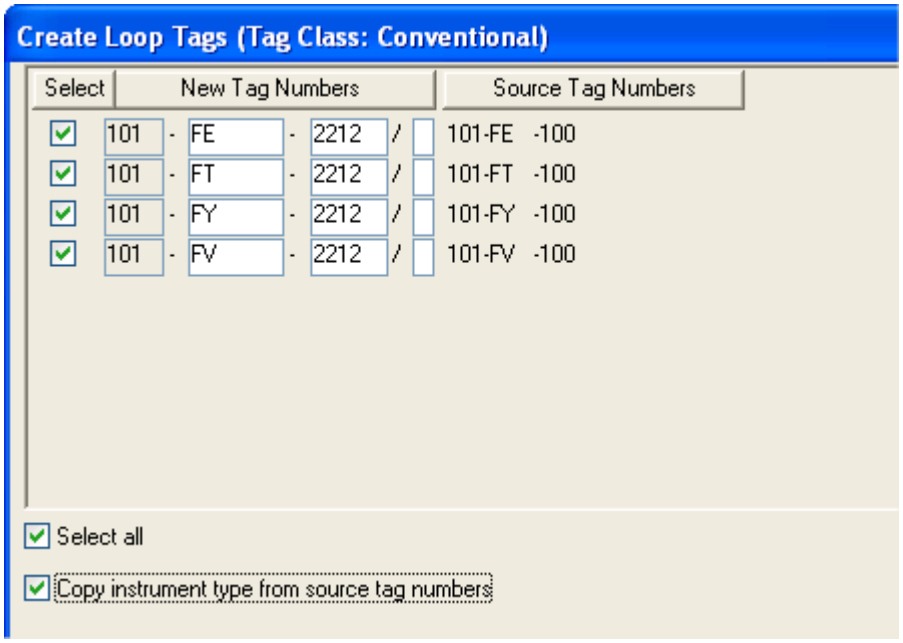


Duplicate a Loop

1. In the **Domain Explorer**, right-click loop **101-F -100**.
2. On the shortcut menu, click **Duplicate**.
3. On the **Duplicated Loop Number** dialog box, change the number of the loop from **100** to **2212**.

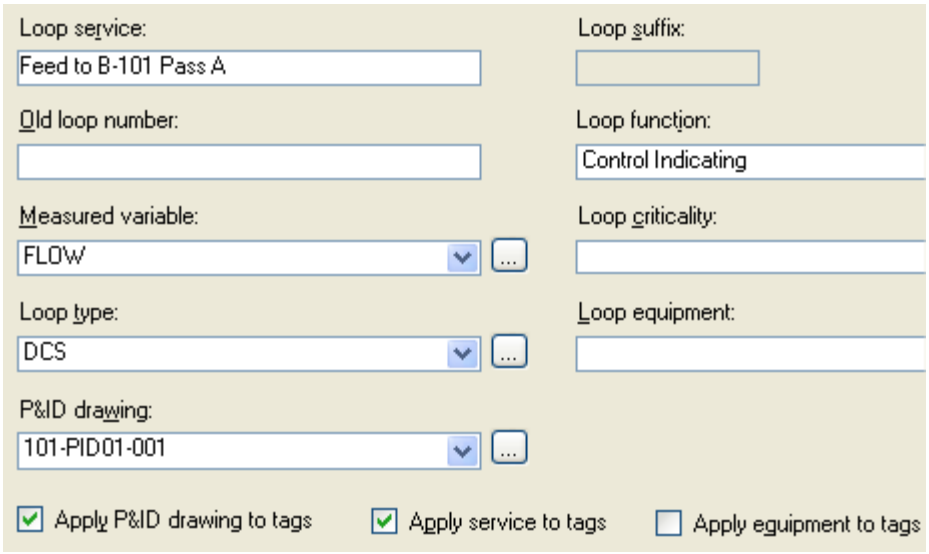


4. Click **OK** to display the **Create Loop Tags** dialog box.
5. Select the **Select all** and **Copy instrument type from source tag numbers** check boxes.
 The dialog box should appear as shown:



6. Click **OK** to start the duplication process.
 At the end of the duplication process, the **Create Loop Tags** dialog box closes and the **Loop Number Properties** dialog box appears.

7. In the **Loop service** field, type **Feed to B-101 Pass A** and make sure that the rest of the information appears as follows.



8. Click **OK** to close the **Loop Number Properties** dialog box.
9. When prompted to edit the tag numbers, click **Yes**.

 The **Tag Number Properties** dialog box opens displaying data of the first tag associated with the new loop (**101-FE -2212**).

 NOTE The P&ID number and loop service have been assigned to the tags automatically because you selected the appropriate check boxes on the **Loop Number Properties** dialog box.

10. Beside **Line**, click

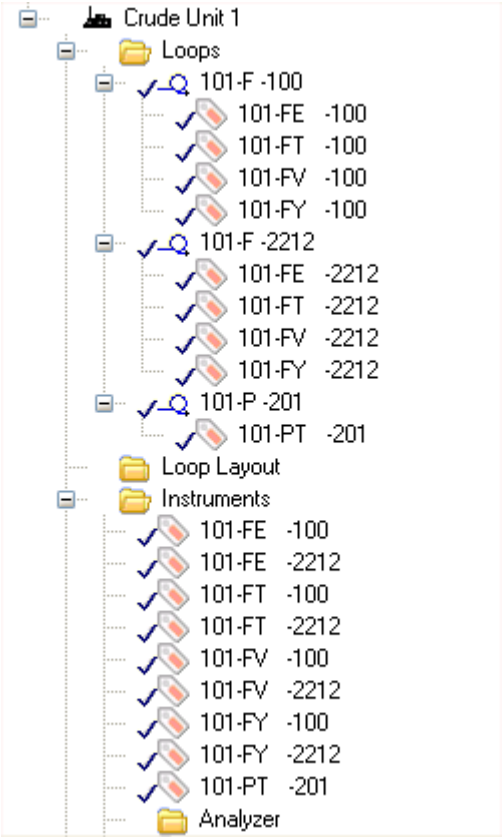
 TIP At the prompt to save data changes, click **Yes**.

11. Add line **3"-FO-1212-4C**.

 TIP When creating line **3"-FO-1212-4C**, define the line number only, without editing the other data fields.

12. Click **OK** to close the **Line Properties (New)** dialog box.
13. Click **OK** to close the **Lines** dialog box.
14. Click **Apply** when you have finished editing the tag data.
15. Click **Next** or **Previous** to move back and forth between the instrument tags associated with the loop.
16. Click **OK** to close the **Tag Number Properties** dialog box.
17. Refresh the items in the **Loops** and **Instruments** folders of the **Domain Explorer**.

18. Expand the **Loops** and **Instruments** folders and make sure that the items appear as shown.



Task 6 — Creating Loops and Tag Numbers in Batch Mode

In this task, you will learn how to create typical loops and tags and then create numerous plant loops in batch mode on the basis of typical loops. A typical loop is a user-defined combination of typical tags that are in fact various instrument types. You create your user-defined typical loop which is then used as a template for batch creation of plant loops in batch mode. This feature can be very useful when you need to create numerous loops based on the same template. You can create as many typical loops as required and use them to create plant loops in batch mode.

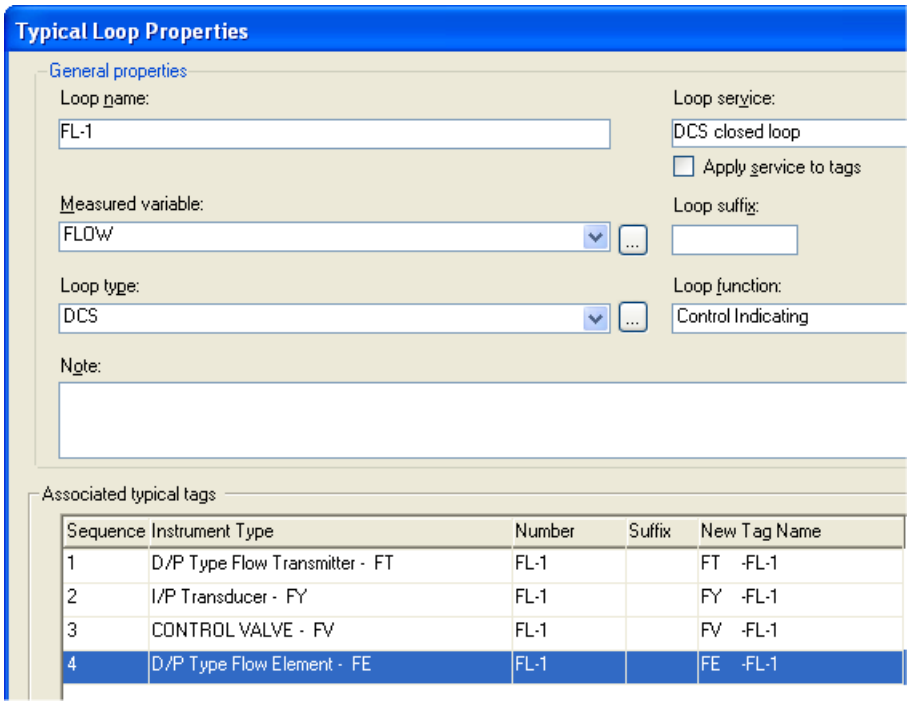
Create Typical Loop FL-1

Creating a typical loop is the first stage in a batch loop creation procedure. At this stage, you make all the typical loop definitions: enter the typical loop name, select its measured variable and loop function. For the typical loop, you can also select the appropriate instrument types to serve as the typical tags.

1. With the **Instrument Index Module** window open, click **Edit** > **Typical Loop Management**.
2. On the **Typical Loop Management** dialog box, click **New** to open the **Typical Loop Properties** dialog box.
3. Enter data as follows:
 a. In the **Loop name** box, type **FL-1**.
 b. From the **Measured variable** list, select **FLOW** as the value for the new typical loop.
 c. From the **Loop type** list, select **DCS**.
 d. From the **Loop function** list, select **Control Indicating** as the loop function for the new typical loop.
 e. In the **Loop service** box, type **DCS closed loop**.
4. In the **Associated typical tags** pane, click **Add** to add a typical tag to the loop.
5. From the **Instrument Type** list, select **D/P Type Flow Element - FE**. This instrument type will serve as a typical tag associated with the current typical loop.

 TIP The **Number** field automatically displays the numeric segment of the typical tag name which is derived from the typical loop name. You can accept it or modify it as needed.
6. Click **Add** to add another typical tag to the current loop.
7. From the **Instrument Type** list, select **D/P Type Flow Transmitter - FT**.
8. Click **Add** again.
9. Add typical tags **I/P Transducer – FY** and **CONTROL VALVE – FV**.

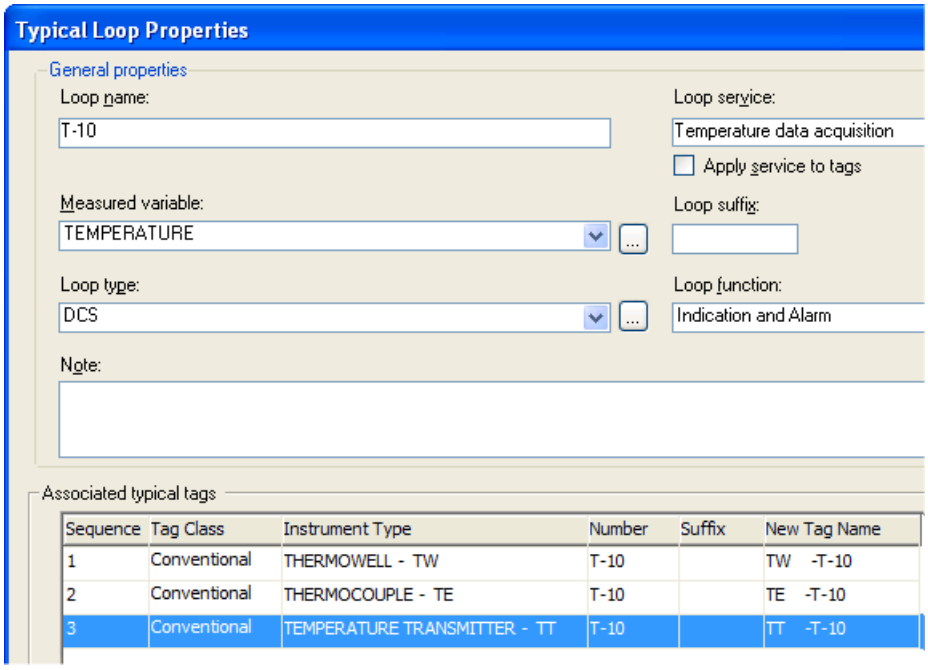
The **Typical Loop Properties** dialog box should appear as shown:



10. Click **OK** and then click **No** when prompted to edit tag number properties.

Create Typical Loop T-10

1. Create a new typical loop **T-10**.
2. Assign properties to the typical loop and associate typical tags as shown.



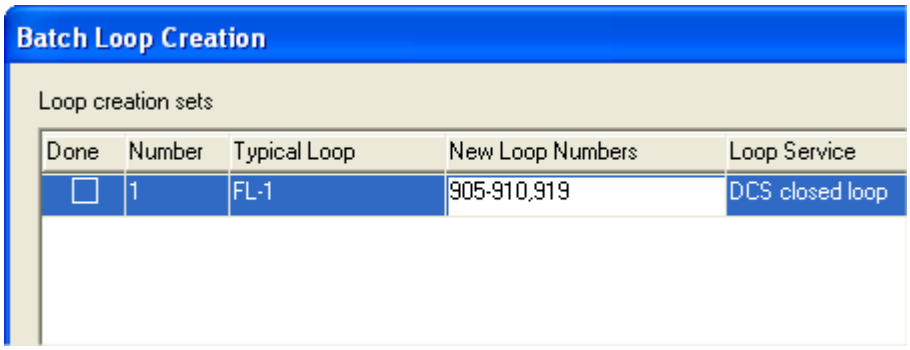
3. After you finish creating typical loop **T-10** with its instrument tags, click **Close** on the **Typical Loop Management** dialog box.

Create Plant Loops in Batch Mode

You will now create several plant loops based on your typical loops.

1. With the **Instrument Index Module** window open, click **Edit** > **Batch Loop Creation**.
2. On the **Batch Loop Creation** dialog box, from the **Typical Loop** list, select **FL-1**.
3. In the **New Loop Number** box, type **905-910, 919** to create loops **F-905** to **F-910**, and also create loop **F-919**.

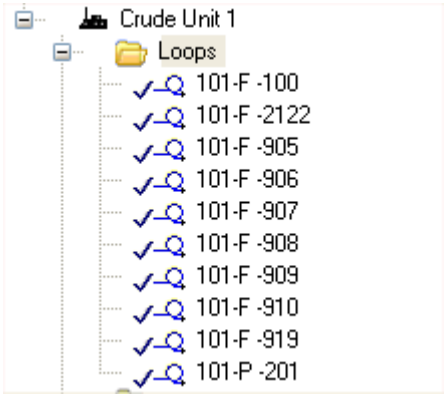
 The dialog box should appear as shown:



TIPS

- You can create as many plant loops based on the selected typical loop as required. To create more than one plant loop, simply type the required new plant loop number names in the **New Loop Number** field. You can enter a range of loop numbers or a list of loop numbers separated by commas.
- For noncontiguous alphanumeric or numeric loop numbers, type the loop numbers with commas between them. For a range of numeric loop numbers, type in the range of loop numbers with a hyphen between them.

4. Click **Create** to start the plant loop creation.
5. After the batch creation process is complete, click **Close** to close the **Batch Loop Creation** dialog box.
6. Open the **Domain Explorer** to view the results.



7. With the **Instrument Index Module** window open, click **Actions > Browse Index** and review the tags and loops that you created.

8. Now create plant loops **T-905** through **T-910** and **T-919** based on typical loop **T-10**.

Task 7 — Creating Additional Loops and Viewing Instrument Index Data

This task enables you to use the various methods of loop and tag creation you have learned.

Create Additional Plant Loops and Instruments

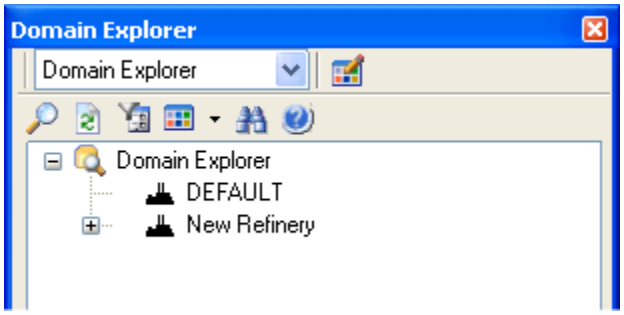
Make sure you create **all** of the following instrument tags as you are going to make use of them in Tutorial tasks later. For full details of the loops and instrument tags, refer to the *Demo P&ID* (on page 14).

Loop Name	Creation Method	Loop service	Instrument Tags	Line
F-102	Duplicate F-100	Feed from C-1	FE, FT	4"-P-1502-11H
F-2213	Duplicate F-2212	Feed to B-101 Pass B	FE, FT, FY, FV	4"-P-1502-11H
F-201	Domain Explorer (as described in *Task 3 — Creating Loops with Tags Using the Domain Explorer* (on page 61))	N/A	FE, FT, FI	N/A
P-208	Domain Explorer (as described in *Task 3 — Creating Loops with Tags Using the Domain Explorer* (on page 61))	N/A	PSH	N/A
L-201	Instrument Index (as described in *Task 4 — Creating a Loop with Its Tag Using the Instrument Index Module* (on page 69))	N/A	LT	N/A
P-100 P-101	Create typical loop P-1 and perform batch loop creation (as described in *Task 6 — Creating Loops and Tag Numbers in Batch Mode* (on page 75)).	N/A	PI (for each loop)	N/A

Search for a Specific Loop and View Loop Tag Properties

In cases where there are a large number of items in the **Domain Explorer**, you can use the **Search** feature to locate a particular item on the tree.

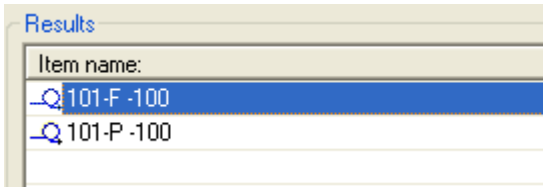
1. In the **Domain Explorer**, collapse the folder hierarchy so that the data appears as follows.



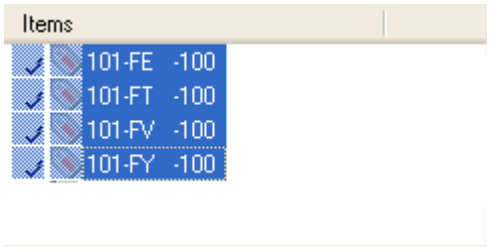
2. In the **Domain Explorer** toolbar, click (**Search**).
3. On the **Search** dialog box, do the following:
 a. From the **Item type** list, select **Loop**.
 b. In the **Item name** box, type ***100***.



 c. Click **Search Now**.
 d. Under **Results**, select loop **101-F -100** as shown.



 e. Click **Go to Item**.
 f. Click **Cancel**.
4. In the **Domain Explorer**, click loop **101-F -100** to display its instrument tags.
5. In the **Items** pane, select all the instruments as shown.

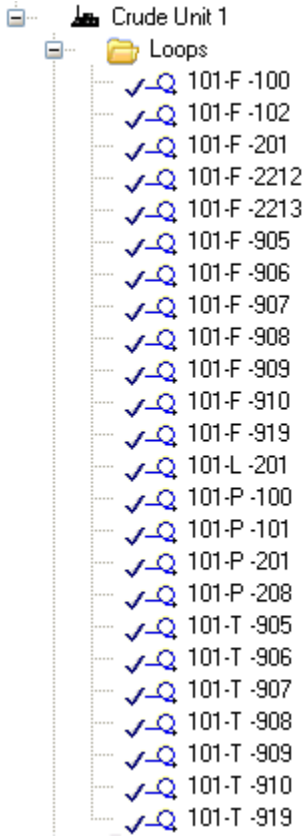


6. Right-click the instruments, and then, on the shortcut menu, click **Properties**.

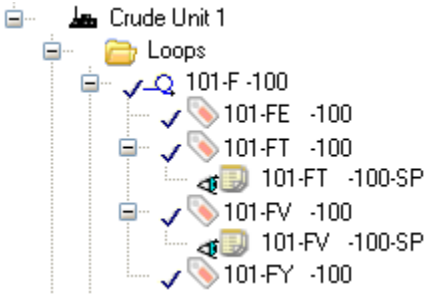
7. On the **Tag Number Properties** dialog box, click **Next** and **Previous** to navigate between instruments and view the properties of each instrument.

View Plant Loops, Instruments, and Associated Items

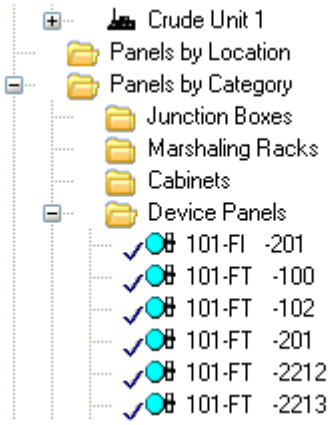
1. In the **Domain Explorer**, refresh the items in the **Loops** folder and view the loops that you created.



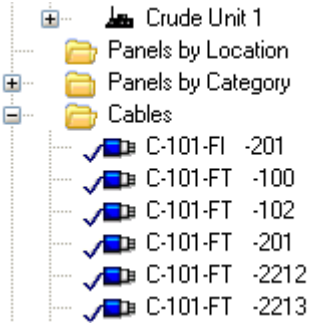
2. Expand the hierarchy of loop **101-F -100** to view its tags and instrument specifications, created automatically according to the instrument type profile of the source **FT-100** and **FV-100** tags.



3. Expand the **Device Panels** folder to view device panels created automatically according to the instrument type profile of the source tags.



4. Expand the **Cables** folder to view the created device cables.



5. In the **Instrument Index Module** window, on the toolbar, click (**Browse**) and view all the instruments that you created.

At this stage, you have completed building your instrument index. You should now feel comfortable with the SmartPlant Instrumentation environment and have a good understanding of the instrument index options.

SECTION 5

Viewing and Modifying Data for Multiple Records

You access and manipulate your data by defining and using browser views. You can customize each browser view to present the data that is important for you – this includes specifying which properties to display, determining the order of the property columns, and sorting or filtering the data records according to criteria that you specify.

Task 1 — Viewing and Editing Instrument Properties from a Browser View

There is a special browser known as the **Instrument Index Standard Browser**, which in addition to allowing you to view and edit tag number data, also provides a means of performing a number of other tasks related to tag numbers, such as generating specification sheets, associating tag numbers with loops, and performing I/O assignments.

View and Edit Data for a Single Tag Number in the Instrument Index Standard Browser

1. Open the Instrument Index module by doing one of the following:
 - On the main toolbar, click .
 - On the main menu bar, click **Modules** > **Instrument Index**.
2. Open the **Browser View** window for the **Instrument Index Standard Browser** by doing one of the following:
 - On the module toolbar, click .
 - Click **Actions** > **Browse Index**.
3. To edit a single data value, click in the cell containing that value.

TIPS

- To move to the next data value in the same row, press the **Tab** key.
- Click **Ctrl** + **Home** to move to the first editable value at the beginning of the tag list or **Ctrl** + **End** to move to the end of the tag list.
- Non-editable data appears with a yellow background.

NOTES

- Some fields only enable you to select from a list. If you want to add to or delete some of the options in a list, you can access the required list from the **Tables** menu in the **Instrument Index Module** window.
- The software automatically saves to the database changes you make to a record when you proceed to another record or when you close the **Browser View** window.

Edit the Browser View Window Records – Multi-Row Editing

Now you will learn how to select a number of tag numbers in the **Instrument Index Standard Browser** for multi-row editing. When selecting the multi-edit option, a dialog box opens where all the available **Browser View** window fields for the selected tag are available. In addition to this multi-editing feature, you are also able to access the appropriate supporting tables if you need to update the available options in the lists.

1. With the **Instrument Index Standard Browser** open, select multiple tags by doing one of the following:
 - To select a group of consecutive tags, select the first tag in the group and then press and hold the **Shift** key, and then click the last tag in the group.
 - To select a group of non-consecutive tags, click the first tag that you want to select, then press and hold the **Ctrl** key as you click the other tags that you want to select. (You can also click **Ctrl** + **Home** to move to the beginning of the tag list or **Ctrl** + **End** to move to the end of the tag list.)
2. Do one of the following:
 - On the **Browse** toolbar, click .
 - Click **Edit** > **Properties** to open the **Tag Number Properties** dialog box.
3. Edit the data as desired.

 TIP To modify data in a supporting table, click the ellipsis button  beside the desired property.
4. Click **Apply** to save all the data for the currently selected tag.
5. Click **Next** to display the next tag to be edited (available only if you selected multiple tags).
6. After making changes, click **Apply** again.
7. When done, click **OK** to return to the **Browser View** window.

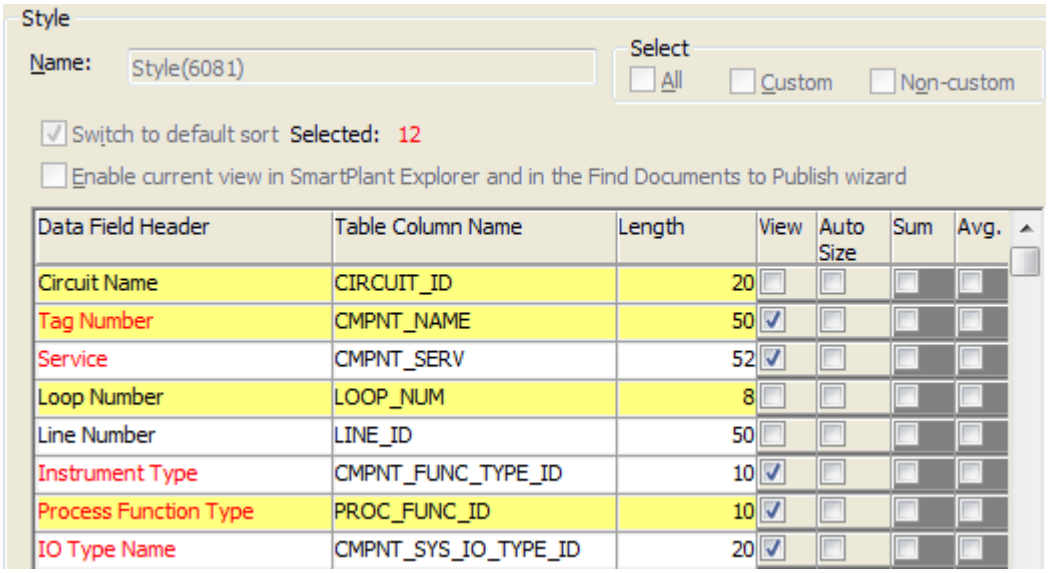
Task 2 — Customizing a Browser View

So far, you have used the **Instrument Index Standard Browser** to display the instrument tag properties with the default view that is shipped with the software. However, you can easily customize the view to display or hide the desired instrument properties. In this task, you will display the custom fields and tables that you created earlier.

Customize the Browser View

1. With the **Instrument Index Standard Browser** open in the **Browser View** window, do one of the following:
 - On the **Browse** toolbar, click  **(Manager)**.
 - Click **Actions** > **Manager**.

2. In the **Browser Manager** window, under the **Browser groups** tree view, under **Instrument Index**, expand the folder hierarchy **Instrument Index Standard Browser** > **Default View** click (**Style**) to display the **Style** section.



Data Field Header	Table Column Name	Length	View	Auto Size	Sum	Avg.
Circuit Name	CIRCUIT_ID	20	☐	☐	☐	☐
Tag Number	CMPNT_NAME	50	☑	☐	☐	☐
Service	CMPNT_SERV	52	☑	☐	☐	☐
Loop Number	LOOP_NUM	8	☐	☐	☐	☐
Line Number	LINE_ID	50	☐	☐	☐	☐
Instrument Type	CMPNT_FUNC_TYPE_ID	10	☑	☐	☐	☐
Process Function Type	PROC_FUNC_ID	10	☑	☐	☐	☐
IO Type Name	CMPNT_SYS_IO_TYPE_ID	20	☑	☐	☐	☐

3. Click **Edit** to enable editing of the style.

 TIP The **Edit** label changes to **Save**.

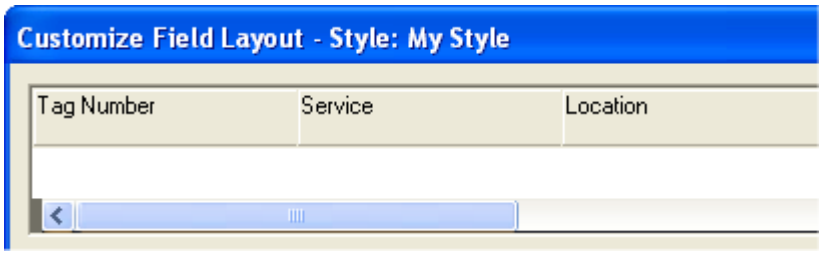
4. In the **Name** field, select the displayed name and overtype this name with a new **style** name — **My Style**.
5. In the **Select** group box, select the **All** check box twice to clear the **View** column for all the selected fields.

 TIP The **Tag Number** property is always selected for displaying because it is the key field.

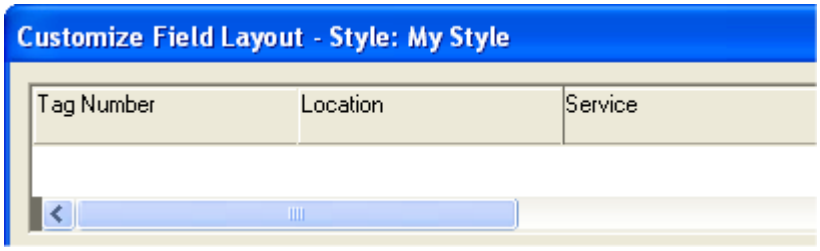
6. Click the **Data Field Header** column header to sort the fields alphabetically in ascending order
7. Select the **View** check box beside the following field names:
 - **Cell number** (custom table)
 - **Contract** (custom field)
 - **Designer** (custom table)
 - **Location**
 - **Lot number** (custom field)
 - **Service**
 - **Supplier** (custom field)
 - **Tag Number** (selected by default)
 - **Technician** (custom table)

 You should now have nine fields selected in the **Style** section.

8. Click **Customize** to open the **Customize Field Layout** dialog box.

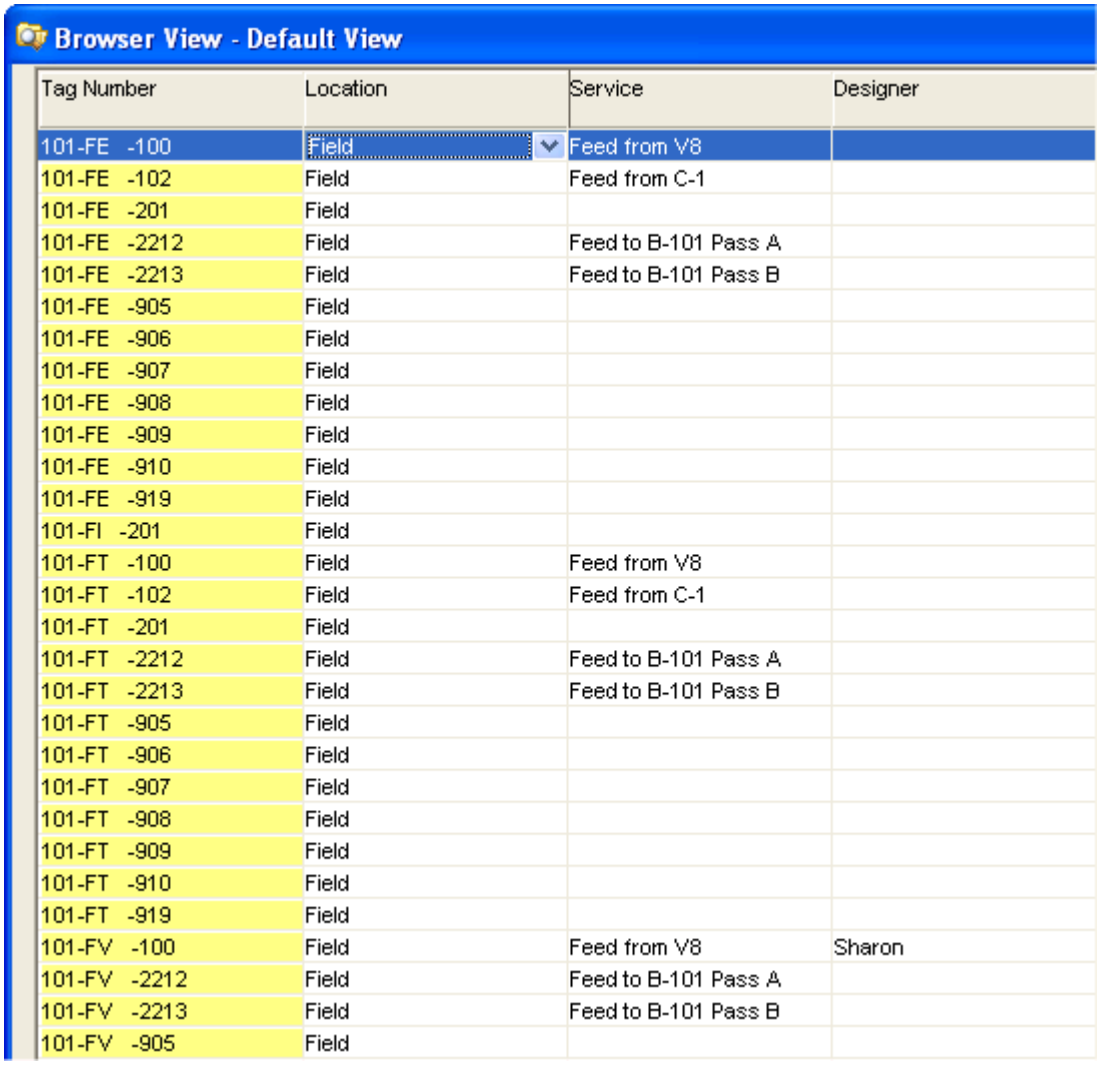


9. Drag the **Location** header to the right of the **Tag Number** header as shown.



10. Click **OK**.

11. On the **Browse** toolbar, click (**Open View**) to display the modified view of the **Instrument Index Standard Browser**.



Browser View - Default View

Tag Number	Location	Service	Designer
101-FE -100	Field	Feed from V8	
101-FE -102	Field	Feed from C-1	
101-FE -201	Field		
101-FE -2212	Field	Feed to B-101 Pass A	
101-FE -2213	Field	Feed to B-101 Pass B	
101-FE -905	Field		
101-FE -906	Field		
101-FE -907	Field		
101-FE -908	Field		
101-FE -909	Field		
101-FE -910	Field		
101-FE -919	Field		
101-FI -201	Field		
101-FT -100	Field	Feed from V8	
101-FT -102	Field	Feed from C-1	
101-FT -201	Field		
101-FT -2212	Field	Feed to B-101 Pass A	
101-FT -2213	Field	Feed to B-101 Pass B	
101-FT -905	Field		
101-FT -906	Field		
101-FT -907	Field		
101-FT -908	Field		
101-FT -909	Field		
101-FT -910	Field		
101-FT -919	Field		
101-FV -100	Field	Feed from V8	Sharon
101-FV -2212	Field	Feed to B-101 Pass A	
101-FV -2213	Field	Feed to B-101 Pass B	
101-FV -905	Field		

TIP You can see the custom table and custom field values that you already entered for tag number **101-FV -100**.

12. Type values in the custom fields and select values from the custom table lists for other tag numbers as desired.

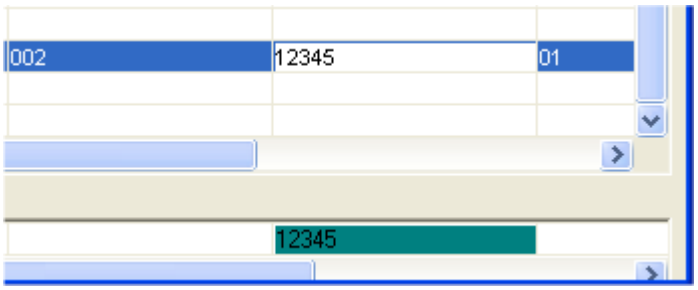
Task 3 — Copying and Pasting Values in the Browser View Window

Copy a Specific Value to Several Fields

This procedure enables you to copy and paste a specific value to one or more fields using the **Ctrl+C** and **Ctrl+V** keys.

1. Reopen the default view of the **Instrument Index Standard Browser**.
2. Double-click the **Tag Number** column header to sort the instruments according to their tag number.

 TIP If you double-click this column header one more time, the software reverses the sort sequence.
3. In the **Contract** column, click the field that displays the **12345** value (in the **101-FV -100** tag number row).
4. Press **Ctrl+C** to display the value in the **Browser View** window buffer.



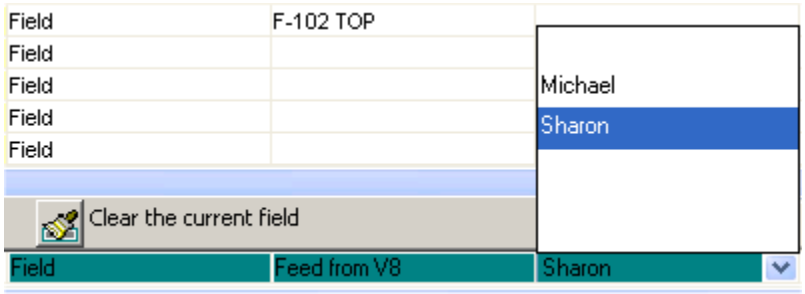
5. In the window, select rows for **101-FV -906** and **101-FV -907** tags.
6. Press **Ctrl+V** to copy the contract value to these instruments.
7. Above the buffer, click (**Clear the buffer**).

Copy Values to Multiple Fields

This procedure enables you to copy and paste multiple values to more than one field using the **Copy** and **Paste** options of the **Browser View** window.

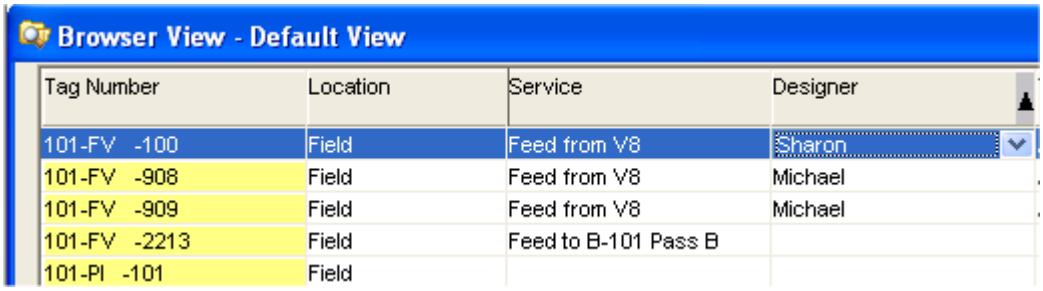
1. In the default view of the **Instrument Index Standard Browser**, click any field in the **101-FV -100** tag number row.
2. Do one of the following to display all the **101-FV -100** field values in the buffer:
 - Click **Edit** > **Copy**.
 - Click on the module toolbar.

3. In the buffer, click the **Designer** column field to display the content of the **Designer** supporting table.



1. Select **Michael**.
2. In the window, select tags **101-FV -908** and **101-FV -909**.
3. Do one of the following to paste the data appearing in the buffer to the selected tags:
 - Right-click and select **Tag Number Activities** > **Paste**.
 - Click on the module toolbar

 CAUTION You should exercise extreme caution when copying data in the **Browser View** window because, unlike the **Ctrl+V** action, the **Paste from Buffer** command copies all the values that appear in the buffer.
4. Above the buffer, click (**Clear the buffer**).
5. Double-click the **Designer** column to sort the data according to designer.
6. The data should appear as shown:



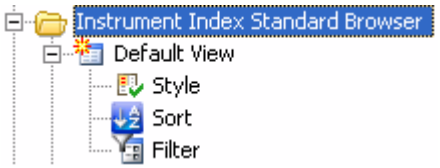
7. Close the **Browser View** window.

Task 4 — Sorting and Filtering Data for the Browser View

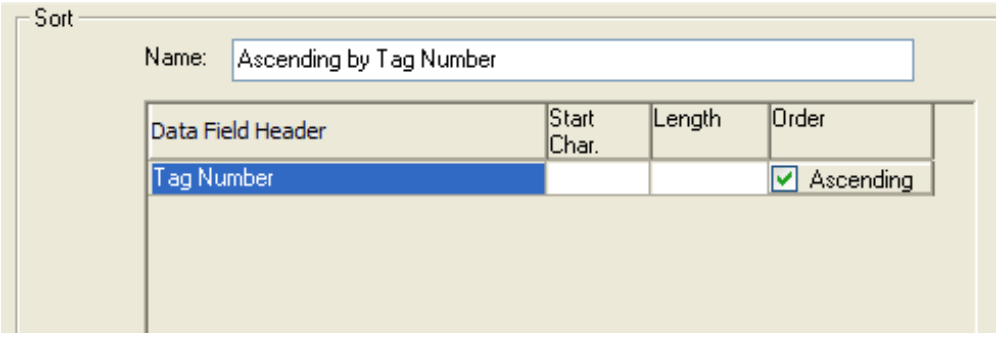
You can now customize the display of your data in the **Instrument Index Standard Browser** by applying the sorting and filtering options to your data.

Create a Customized Sorting Sequence

1. Make sure that the hierarchy tree of the **Instrument Index Standard Browser** is expanded. If it is not, double-click it so that it appears as shown.



2. In the **Browser groups** pane, click **Sort** to display the **Sort** section.
3. Click **Edit** to open the fields for editing.
4. In the **Name** box, type **Ascending by Tag Number**.
5. Click the highlighted **Data Field Header** column and, from the list, select **Tag Number**.
6. In the **Order** column, select the **Ascending** check box.



7. Click **Save** to save your sort definition.

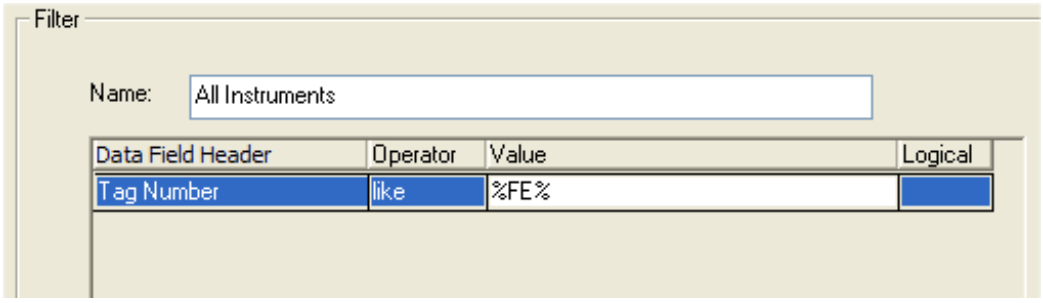
 Your new sort definition now appears in the **Sort Field List** pane in the **Browser Manager**.

Create a Customized Filter

The Browser module enables you to define a filter for your new browser. However, the browser you are creating now does not require a filter.

1. Make sure that the hierarchy tree of the **Instrument Index Standard Browser** is expanded. If it is not, double-click it.
2. In the **Browser groups** pane, click **Filter** to display the **Filter** section.
3. Click **Edit**.
4. In the **Name** box, type **All Instruments**.
5. Define the filter criteria as follows:
 a. In the **Data Field Header** column, select **Tag Number**.
 b. In the **Operator** column, select **like**.
 c. In the **Value** column, type **%FE%**.

NOTE The **%** symbol behaves as a wildcard, and in this example, allows you to display only those tag numbers that contain the **FE** string.

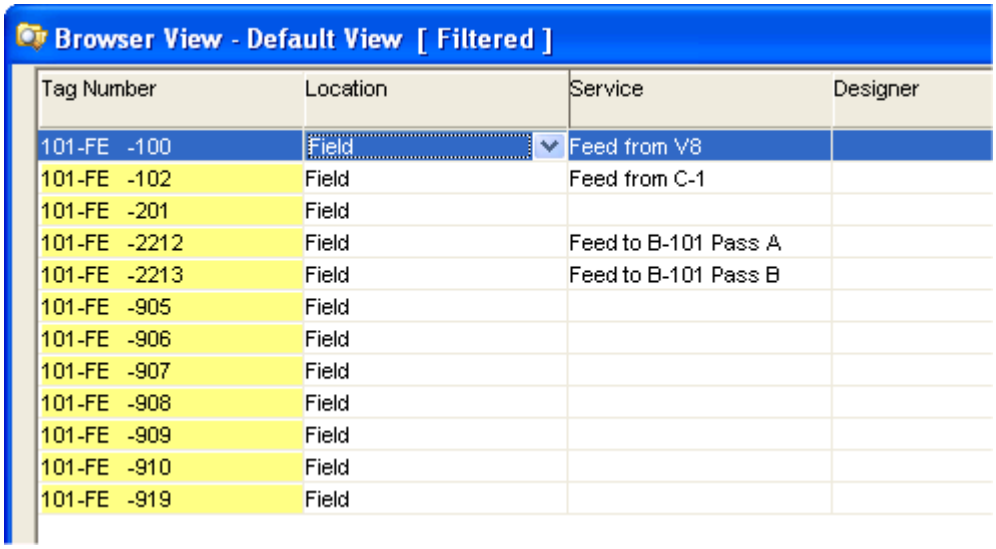


6. Click **Save**.

Open the Sorted and Filtered Browser

Now that you have applied the sorting and filter criteria to your data, you can open it in the browser view.

1. Under the **Instrument Index Standard Browser**, select **Default View**.
2. On the **Browser** toolbar, click (**Open View**) to open the filtered browser view.



3. If desired, modify the layout on-the-fly as follows:
 a. To change the column sequence, drag column headers to the new positions where you want them to appear.
 b. To change the width of a column, position the cursor on the border between two column headers and drag the border to the right or to the left.

TIP If you change the layout from the browser view, to retain the modified column sequence and widths the next time you open the browser view, right-click anywhere inside the browser view, and on the shortcut menu, click **Save Field Layout**.

SECTION 6

Defining Process Data

In this section, you will create lines, define line process data, and then associate this data with the instruments located on these lines. You can then use this process data when performing calculations and generating specifications.

Task 1 — Defining Process Data for Lines

This task involves creating a line in the Process Data module, defining process data for this line and copying the process data to another line.

Create a Line

At this stage, it is assumed that you have already created lines **4"-P-1501-11H** and **4"-P-1502-11H** when defining you instrument index data. Now you are going to create a new line in the Process Data module.

1. To open the Process Data module, do one of the following:
 - On the main toolbar, click .
 - On the **Modules** menu, click **Process Data**.
2. On the module toolbar, click (**Line**).
3. On the **Select Line** dialog box, from the **Line Type** list, select **Utilities**.
4. Click **New** to open the **Line Properties** dialog box.
5. Define new line **1.5"-S-2001-4C** and click **OK** to return to the **Select Line** dialog box.
6. On the **Select Line** dialog box, click **OK** again to complete the new line creation process.
7. On the **Process Data** dialog box, from the **Fluid state** list, select **Liquid**, and then, click **OK**.

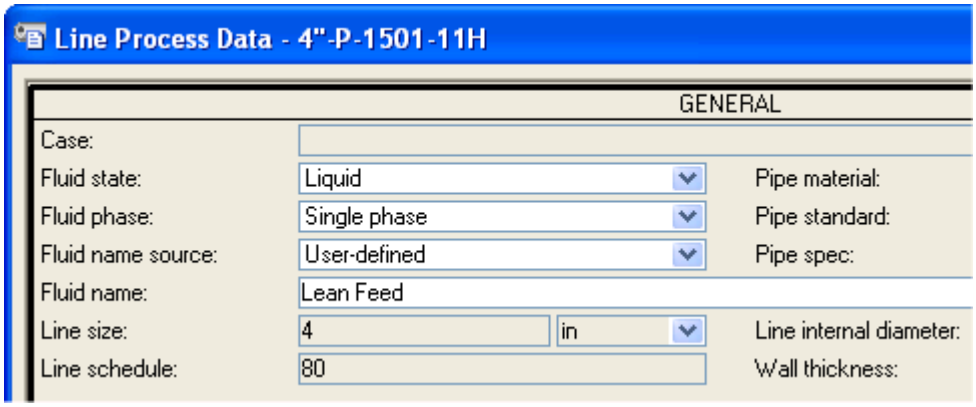
Define Line Process Data

1. On the toolbar, click (**Line**).
2. On the **Select Line** dialog box, select the **Show all line types** check box.
 Your line data should appear as shown:

Line Number	P&ID Number	Stream Name	PD Exists
1.5"-S-2001-4C			No
3"-FO-1212-4C			No
3"-FO-1213-4C			No
4"-P-1501-11H			No
4"-P-1502-11H			No

3. Select line number **4"-P-1501-11H** and click **OK**.
4. On the **Process Data** dialog box, from the **Fluid state** list, select **Liquid**, and then, click **OK**.

5. In the **Line Process Data - 4"-P-1501-11H** window, in the **GENERAL** section, enter the values for the fluid phase and fluid name values as shown.



6. In the **PROPERTIES** section, enter the property values in metric units of measure, as shown.

 TIP You must first modify the Units values, before entering the numerical values in **@Minimum**, **@Normal**, and **@Maximum**

PROPERTIES

Report flags:	Density				
	@Minimum	@Normal	@Maximum	Units	
Volumetric flow:	25	30	32	m³/h	@flow
Pressure:	12	13	14	bar	gage
Temperature:	150	150	150	°C	
Viscosity:	0.1	0.1	0.1	cP	
Velocity	0.936	1.12	1.2	m/s	
Density:	890	890	890	kg/m³	
Specific gravity:	0.891	0.891	0.891		
Compressibility:					
Specific heat:	1	1	1	BtuIT/lb°F	
Specific heat ratio:					
Vapor pressure:	0.9	0.9	0.9	bar	gage
Critical pressure:	1200			bar	gage

7. On the toolbar, click (**Save**).
8. On the toolbar, click (**Report**)
9. When prompted to preview the report, click **Yes** to open the print preview of the line process data report.
10. On the toolbar, click (**Revisions**).
11. On the **Revisions** dialog box, enter the revision data as follows:
 a. From the **Revision method** list, select **0,1,2…**

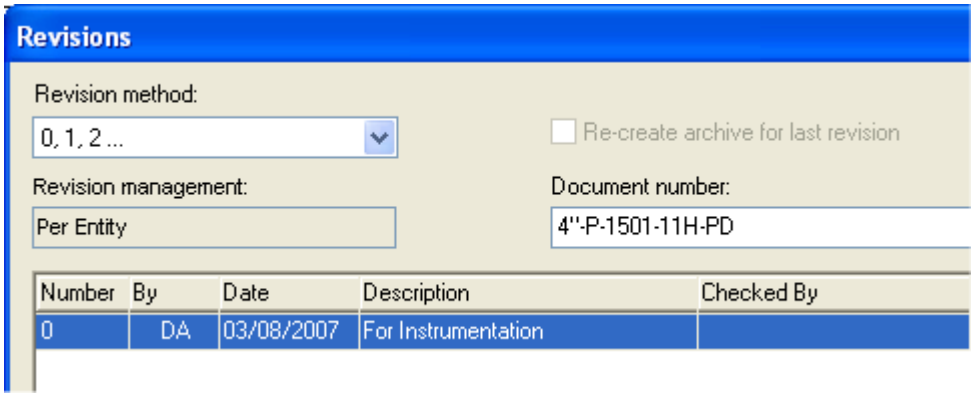
 TIP When you first select a revision numbering method, several options are available to you, including preliminary revisions (designated by **P0, P1, P2**…). Once you select one of the other revision methods, you will not be able to return to the preliminary revision method and this option will be disabled.

 b. Click **New** to enter a revision row with the **O** (zero) revision number and the current date.

c. In the **By** field, enter your initials.

d. In the **Description** field, type **For Instrumentation**.

The dialog box should appear as shown:



12. Click **OK** to close the **Revisions** dialog box.

 TIP After you have made several revisions, you can update them in batch mode. In the main window of any SmartPlant Instrumentation module, click **Tools** > **Global Revisions**. For an example, see *Make Global Revisions for Specifications* (on page 124).

13. Close the print preview window.
14. Close the **Line Process Data** window.
15. On the toolbar, click (**Line**) to reopen the **Select Line** dialog box.

 In the **PD Exists** column, the software now indicates that process data exists for line **4"-P-1501-11H**.

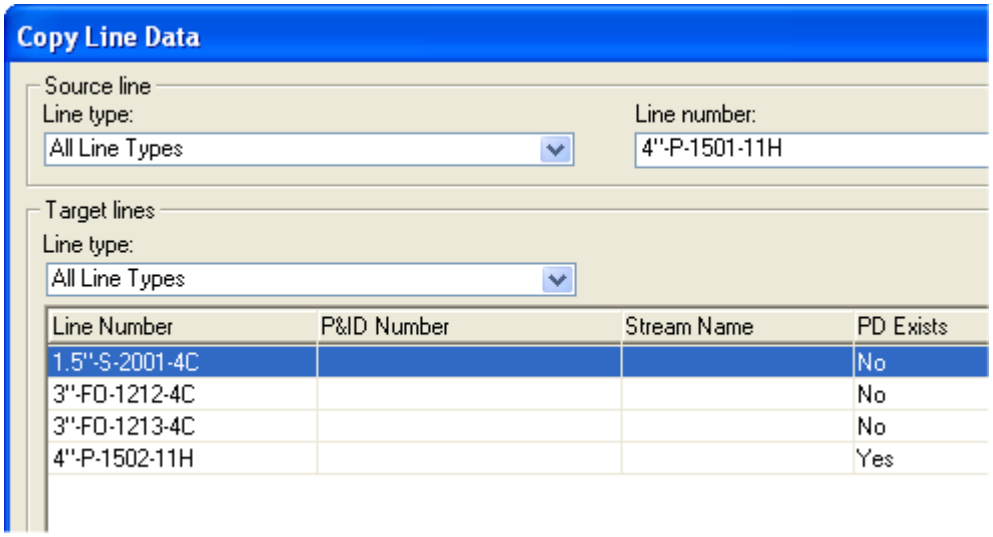
Line Number	P&ID Number	Stream Name	PD Exists
1.5"-S-2001-4C			No
3"-FO-1212-4C			No
3"-FO-1213-4C			No
4"-P-1501-11H			Yes
4"-P-1502-11H			No

16. Click **Cancel** to close the **Select Line** dialog box.

Copy Process Data to Another Line

1. In the **Process Data Module** window, on the menu bar, click **Actions** > **Copy Line Data**.
2. Under **Source line**, make sure that the **Line number** box displays **4"-P-1501-11H**.
3. Under **Target lines**, select line **4"-P-1502-11H**.
4. Click **Copy**.

The **Copy Line Data** dialog box should now indicate that process data exists for line **4"-P-1501-11H:**



5. Click **Close**.

Task 2 — Defining Process Data for Instruments

Now you will define process data for a flowmeter and control valve. Also, you will generate process data reports and make report revisions. These procedures are sufficient to continue with the tasks dealing with calculation and sizing of instruments.

Define Process Data for a Flowmeter

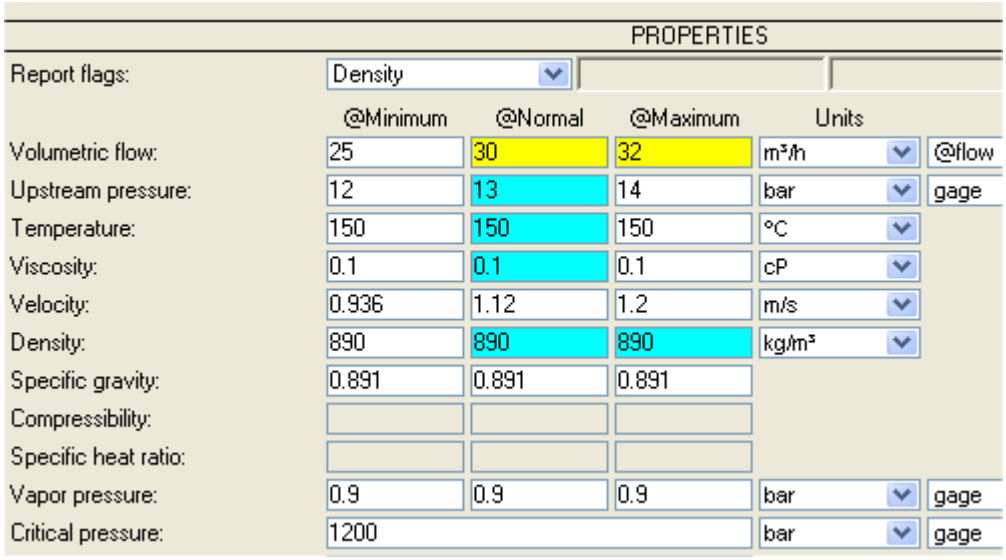
In this procedure, you use the Process Data module options to define process data for instrument tag **101-FE -100** by copying process data from line **4"-P-1501-11H**.

1. On the module toolbar, click **Instrument** .
2. On the **Enter Tag Number** dialog box, type the instrument tag number **101-FE -100** and click **OK**.

 TIP Use the **Tab** key to move the cursor to the character separators.
3. After the **Process Data** window opens, click **Edit** > **Copy From** > **Line**.
4. Click **OK** to copy data from line **4"-P-1501-11H** to the current flowmeter and open the **Flowmeter Process Data** window.

 TIP The software allows you to copy process data from a line on the fly in this way because you already assigned line **4"-P-1501-11H** to tag number **101-FE -100** when defining the tag number properties.

The **PROPERTIES** section should appear as shown:

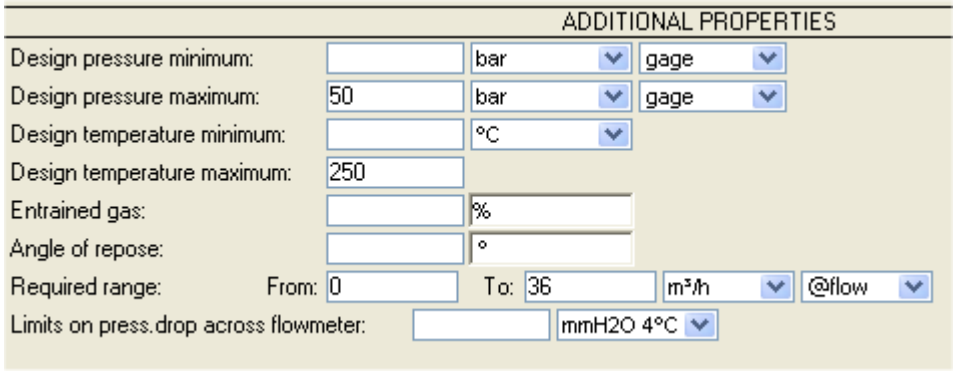


PROPERTIES

Report flags:	Density				
	@Minimum	@Normal	@Maximum	Units	
Volumetric flow:	25	30	32	m³/h	@flow
Upstream pressure:	12	13	14	bar	gage
Temperature:	150	150	150	°C	
Viscosity:	0.1	0.1	0.1	cP	
Velocity:	0.936	1.12	1.2	m/s	
Density:	890	890	890	kg/m³	
Specific gravity:	0.891	0.891	0.891		
Compressibility:					
Specific heat ratio:					
Vapor pressure:	0.9	0.9	0.9	bar	gage
Critical pressure:	1200			bar	gage

TIP Fields that are mandatory for calculations appear with either a cyan or yellow background. Cyan indicates that you **must** enter a value on the process data sheet. Yellow indicates that you may enter a value on the process data sheet, or you can leave it blank and enter a value for that field later on the calculation sheet.

5. Scroll down to display the **Additional Properties** section, and enter the values as shown.



ADDITIONAL PROPERTIES

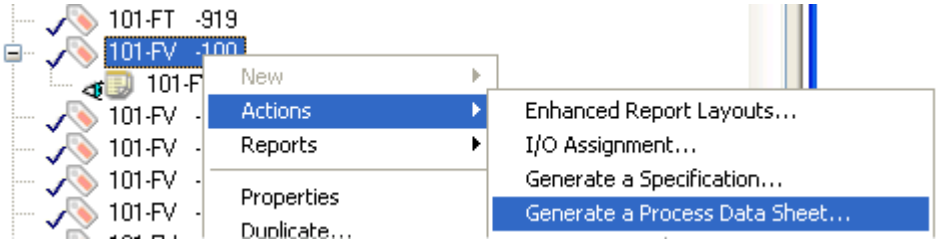
Design pressure minimum:		bar	gage			
Design pressure maximum:	50	bar	gage			
Design temperature minimum:		°C				
Design temperature maximum:	250					
Entrained gas:		%				
Angle of repose:		°				
Required range:	From: 0		To: 36	m³/h	@flow	
Limits on press.drop across flowmeter:		mmH2O 4°C				

6. On the toolbar, click (**Save**).
7. On the toolbar, click (**Report**)
8. Click **Yes** when prompted to open the report print preview.
9. On the toolbar, click (**Revisions**).
10. On the **Revisions** dialog box, enter the revision data. For details, see *Define Process Data for a Line* (see "*Define Line Process Data*" on page 93).

Define Process Data for a Control Valve

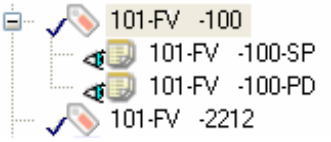
In this procedure, you use the **Domain Explorer** to open a process data sheet for instrument tag **101-FV -100**. Then, you copy process data from flowmeter **101-FE -100** and modify the values for the control valve.

1. Press **F7** to open the **Domain Explorer**.
2. Expand the hierarchy **New Refinery** > **Crude Area** > **Crude Unit 1** > **Instruments**.
3. Right click **101-FV -100**.
4. On the shortcut menu, click **Actions** > **Generate a Process Data Sheet**.



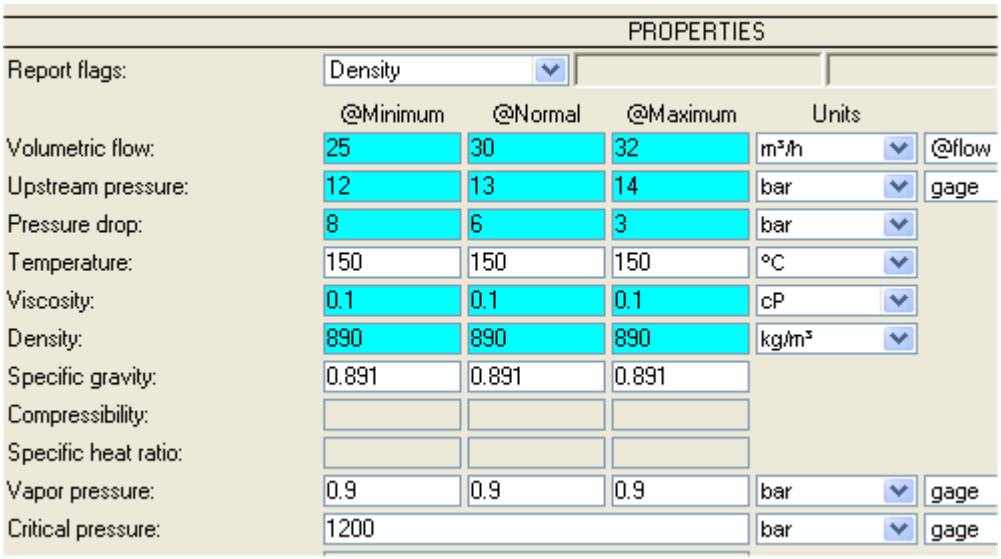
5. On the **Results** dialog box, check whether the **Result Description** column indicates that the process data sheet has been generated successfully, and then, click **OK**.
6. On the **Domain Explorer** toolbar, click (**Refresh**).

 Process data document **101-FV 100-PD** should now appear under **101-FV -100**, in addition to spec document **101-FV 100-SP**, which was created automatically according to the instrument type profile definitions:



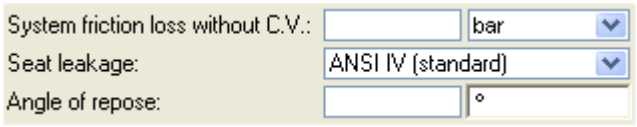
7. Right-click **101-FV 100-PD**.
8. On the shortcut menu, click **Actions** > **Open Document** to open the **Control Valve Process Data** window.
9. Close the **Domain Explorer** to maximize the **Control Valve Process Data** window.
10. On the menu bar, click **Edit** > **Copy From** > **Instrument**.
11. On the **Copy From Instrument** dialog box, type **101-FE -100**, and then, click **OK**.

12. In the **Properties** section, define the pressure drop and upstream pressure values and ensure that the other values appear as shown.



PROPERTIES						
Report flags:	Density					
	@Minimum	@Normal	@Maximum	Units		
Volumetric flow:	25	30	32	m³/h	@flow	
Upstream pressure:	12	13	14	bar	gage	
Pressure drop:	8	6	3	bar		
Temperature:	150	150	150	°C		
Viscosity:	0.1	0.1	0.1	cP		
Density:	890	890	890	kg/m³		
Specific gravity:	0.891	0.891	0.891			
Compressibility:						
Specific heat ratio:						
Vapor pressure:	0.9	0.9	0.9	bar	gage	
Critical pressure:	1200			bar	gage	

13. Scroll down to display the **Additional Properties** section, and specify the following control valve properties.



System friction loss without C.V.:		bar
Seat leakage:	ANSI IV (standard)	
Angle of repose:		°



Failure action:	Close
Handwheel:	No

14. Generate a control valve process data report and add a revision as you have previously done for the flowmeter process data report.

SECTION 7

Performing Calculations and Sizing

Using the process data that you have entered you can now calculate the bore of the orifice plate **101-FE -100** and size your control valve **101-FV -100**. You perform these procedures in the Calculation module.

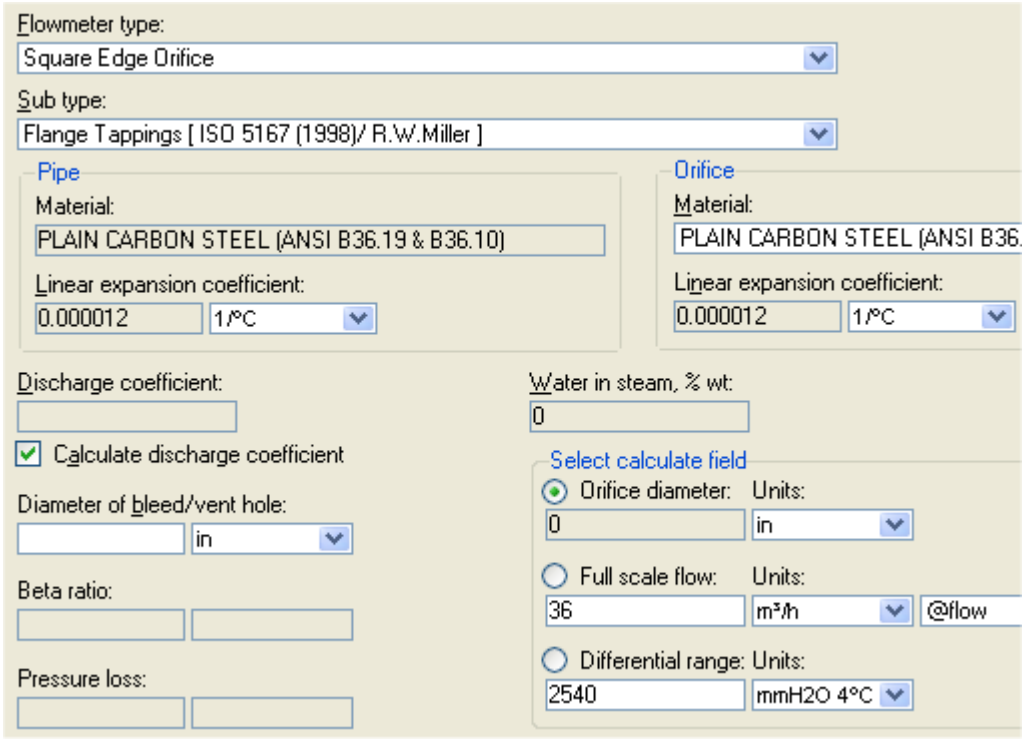
Task 1 — Calculating Flowmeter Parameters

Calculate Flowmeter Parameters

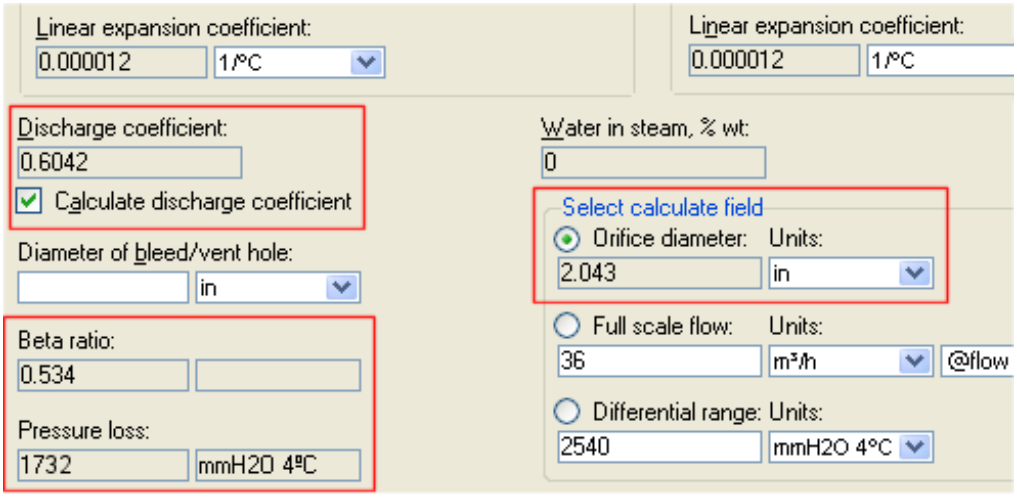
1. To open the Calculation module, do one of the following:
 - On the main toolbar, click .
 - On the main menu bar, click **Modules** > **Calculation**.
2. On the module toolbar, click  (**Flowmeter**).
3. On the **Enter Tag Number** dialog box, click **Find**.
4. On the **Find Tag** dialog box, click **Find**.
5. Select tag number **101-FE -100** and click **OK**.
6. On the **Enter Tag Number** dialog box, click **OK**.

 TIP The calculation sheet for the selected instrument is displayed with instrument process data already populated.
7. On the module toolbar, click  (**Calculate**) to open the **Flowmeter Calculation** dialog box, where you can calculate the orifice diameter.
8. On the **Flowmeter Calculation** dialog box, do the following:
 a. Under **Orifice**, from the **Material** list, select **PLAIN CARBON STEEL (ANSI B36.19 & B36.10)**.
 b. Under **Select calculate field**, select **Orifice diameter** as the parameter to be calculated.

c. Enter the differential range as shown.



9. Click **Calculate** to perform the calculation.
 On completion of the calculation, you should obtain the following results:



10. When done, click **Close** to return to the **Flowmeter Calculation** window.
11. On the module toolbar, click (**Save**).
12. At this stage, you are going to make a back calculation for this bore (**2.051"**) for a new constant of **0-40 m3/h** as follows:
 a. On the toolbar, click to open the **Flowmeter Calculation** dialog box with the **Orifice diameter** value set as **0** (zero).

b. Click **Calculate** to rerun the calculation so that a value for the orifice diameter now appears.
c. Under **Select calculate** field, select **Differential range**.
d. In the **Full scale flow** box, enter **40**, and from the **Units** list, select **m3/h** as the full-scale flow unit of measure.
e. Click **Calculate** to calculate the differential pressure.

A result of **3138 mmH2O 4°C** should be obtained.

13. Close the **Flowmeter Calculation** dialog box to return to the **Flowmeter Calculation** window.
14. Click (**Report**) to generate and preview the calculation report.
15. Click and enter a revision. For details, see *Define Process Data for a Line* (see "*Define Line Process Data*" on page 93).

Task 2 — Calculating and Sizing Control Valve Parameters

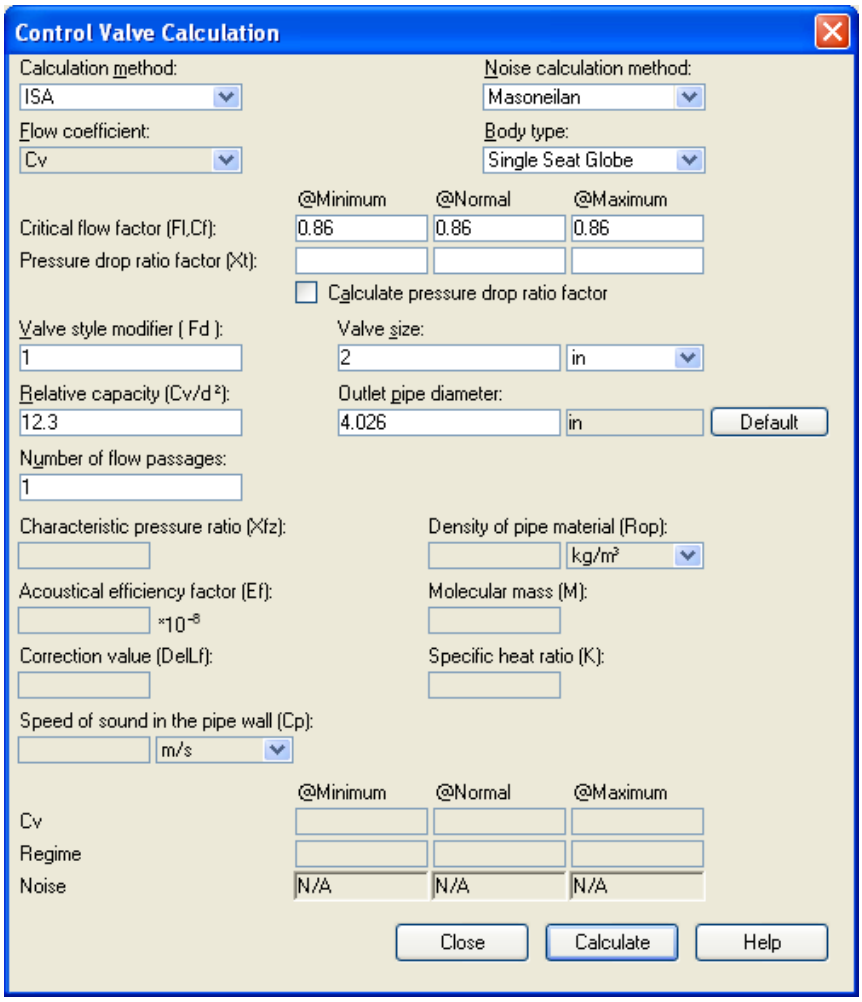
Calculate and Size Control Valve Parameters

1. On the module toolbar, click (**Control Valve**).
2. On the **Enter Tag Number** dialog box, type the instrument tag number **101-FV -100** and click **OK** to display the calculation sheet for the control valve instrument.
3. Ensure that the **Line** box displays **4"-P-1501-11H** and all highlighted fields in the **Properties** section contain data as shown.

PROPERTIES				
Report flags:	Density			
	@Minimum	@Normal	@Maximum	Units
Volumetric flow:	25	30	32	m³/h
Upstream pressure:	12	13	14	bar
Pressure drop:	8	6	3	bar
Temperature:	150	150	150	°C
Viscosity:	0.1	0.1	0.1	cP
Density:	890	890	890	kg/m³
Specific gravity:	0.891	0.891	0.891	
Compressibility:				
Specific heat ratio:				
Vapor pressure:	0.9	0.9	0.9	bar
Critical pressure:	1200			bar

4. On the toolbar, click to open the **Control Valve Calculation** dialog box where you calculate and size the control valve.

5. Enter data as shown:



6. Click **Calculate** to perform the calculation.

 The results are displayed on completion of the calculation as shown:

	@Minimum	@Normal	@Maximum
Cv	9.68	13.5	20.4
Regime	Incip.-Turb.	Turbulent	Turbulent
Noise	77.9	69.4	65.2

7. When done, click **Close** to return to the **Control Valve Calculation** window.
8. On the toolbar, click .
9. Click  to generate and preview the calculation report.
10. Click  to open the **Revisions** dialog box, and enter a revision. For details, see *Define Process Data for a Line* (see "*Define Line Process Data*" on page 93).

You have now successfully completed the calculations required for this tutorial. The software will subsequently use the calculation results in the specification sheets of these instruments.

SECTION 8

Working with Specifications

A specification in SmartPlant Instrumentation is a document that incorporates data from the Instrument Index, Process Data, and Calculation modules. When a spec form is assigned to a particular instrument type profile, the software generates a specification automatically, on creating a tag number belonging to this instrument type profile.

Generated specifications automatically appear in the **Domain Explorer**. You can open an existing spec either from the **Domain Explorer**, Specifications module, or Instrument Index Standard Browser view.

Every specification is based on a spec form, which, in turn, can contain one or more data pages and a note page. SmartPlant Instrumentation is delivered with a library of predefined spec forms with pages.

Task 1 — Generating, Viewing, and Editing Specifications

At this stage, we assume that you have already entered your instrument data, assigned process data to the flow instruments, and calculated the orifice plate and control valves.

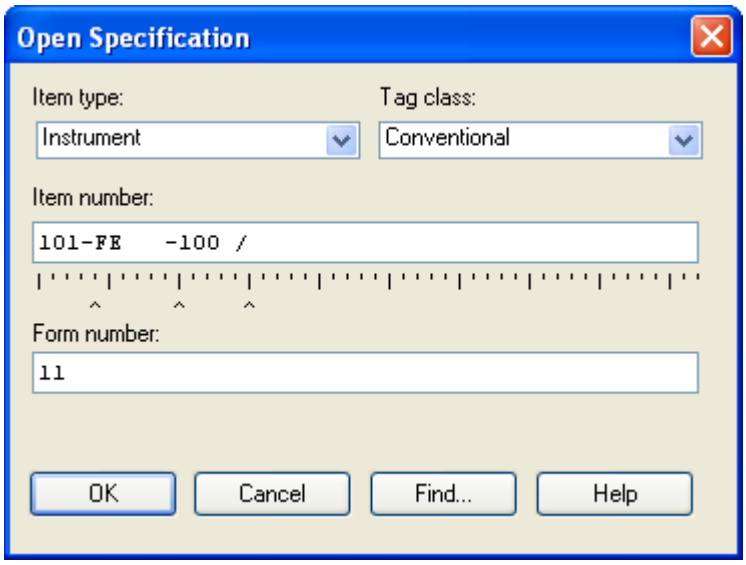
In this task, you will generate a single-tag spec for flowmeter **101-FE -100** and add some data. Also, you will open an existing single-tag spec for control valve **101-FV -100** and modify some values. In the current domain, when you created the **101-FV -100** tag, the software generated a spec automatically, according to the FV Control Valve instrument type profile settings associated with this tag.

Generate a Specification for a Flowmeter

In this procedure, you use the Specifications module options to generate an instrument specification for the tag **101-FE -100** and define orifice flange properties.

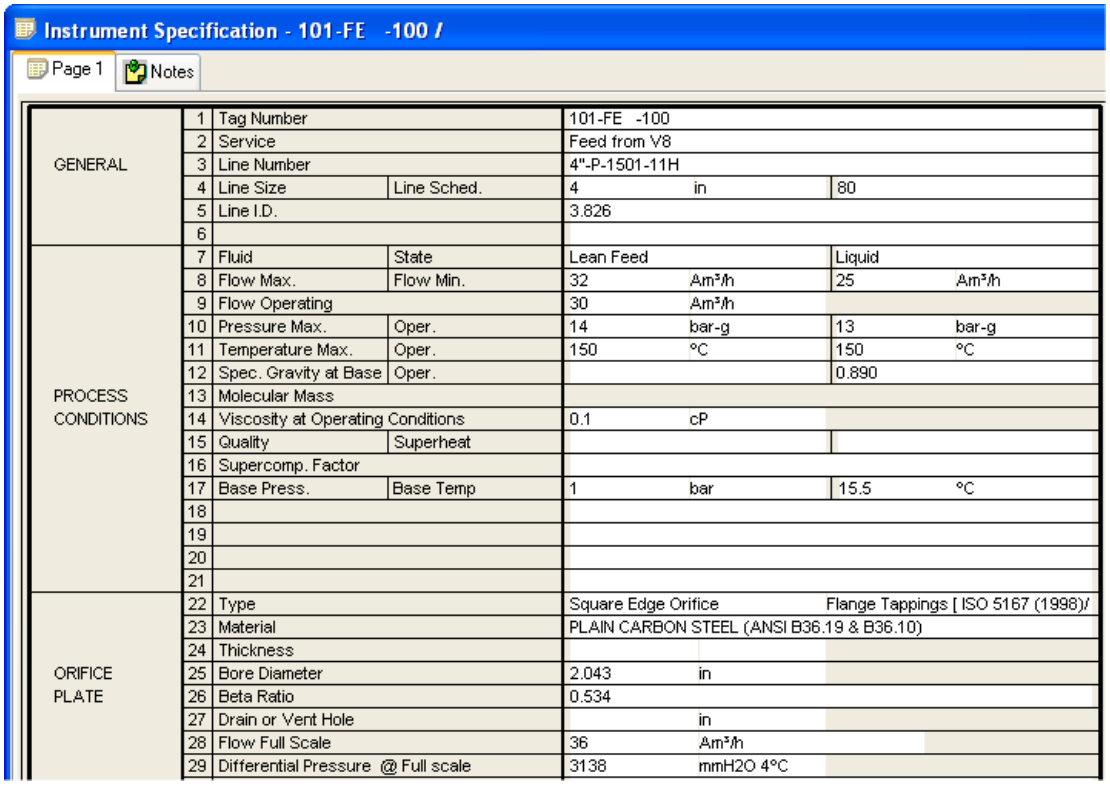
1. To open the Specifications module, do one of the following
 - On the main toolbar, click .
 - On the **Modules** menu, click **Specifications**.
2. On the **Specifications Module** window toolbar, click  (**Open Spec**).
3. On the **Open Specification** dialog box, in the **Item number** box, enter **101-FE -100**.
4. Place the cursor in the **Form number** box and click **Find**.
5. On the **Select Specification Form** dialog box, select form **11 Orifice Plate**, and then, click **OK**.

The information is now displayed as shown:



6. Click **OK**.
7. On the **New Specification** dialog box, click **OK** to generate a specification and open the **Instrument Specification** window.

 This spec sheet already contains data (Service, Line Number, process conditions, and so forth) that was already entered or calculated in the previous tasks:



GENERAL	1	Tag Number		101-FE -100			
	2	Service		Feed from V8			
	3	Line Number		4"-P-1501-11H			
	4	Line Size	Line Sched.	4	in	80	
	5	Line I.D.		3.826			
	6						
PROCESS CONDITIONS	7	Fluid	State	Lean Feed		Liquid	
	8	Flow Max.	Flow Min.	32	Am³/h	25	Am³/h
	9	Flow Operating		30	Am³/h		
	10	Pressure Max.	Oper.	14	bar-g	13	bar-g
	11	Temperature Max.	Oper.	150	°C	150	°C
	12	Spec. Gravity at Base	Oper.			0.890	
	13	Molecular Mass					
	14	Viscosity at Operating Conditions		0.1	cP		
	15	Quality	Superheat				
	16	Supercomp. Factor					
	17	Base Press.	Base Temp	1	bar	15.5	°C
	18						
	19						
	20						
	21						
ORIFICE PLATE	22	Type		Square Edge Orifice		Flange Tappings [ISO 5167 (1998)/	
	23	Material		PLAIN CARBON STEEL (ANSI B36.19 & B36.10)			
	24	Thickness					
	25	Bore Diameter		2.043	in		
	26	Beta Ratio		0.534			
	27	Drain or Vent Hole			in		
	28	Flow Full Scale		36	Am³/h		
	29	Differential Pressure @ Full scale		3138	mmH2O 4°C		

8. Enter additional data in the **Orifice Flanges** section as shown.

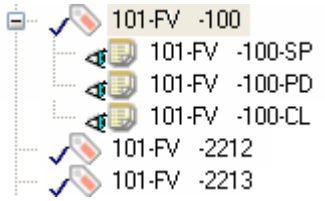
ORIFICE FLANGES	31	Size	4 in
	32	Rating	300#
	33	Type	WELD NECK
	34	Facing	RF
	35	Flange Material	By piping
	36	Gasket Material	
	37	Taps Type	
	38	Taps Size	
	39	Taps Orientation	

9. Click the **Notes** tab and type the text: **1. Orifice flange assembly to be provided by piping**.
10. On the toolbar, click  to save the specification sheet.
11. On the toolbar, click  to refresh the specification sheet.
12. Click the **Page 1** tab and view the note that you typed in the **Notes** section of the specification sheet.
13. Click  to close this specification sheet and return the **Specifications Module** window.

Open a Specification for a Control Valve

In this procedure, you use the **Domain Explorer** to open an existing specification for the **101-FV -100** control valve. This specification has been generated automatically according to the settings of the FV Control Valve instrument type profile.

1. In the **Domain Explorer**, double-click the **101-FV -100** instrument to display the document numbers of the existing documents.



TIP For the displayed documents, document numbers have been created automatically when you generated the documents. By default, such document numbers have the following naming convention:

<source instrument tag> + document type suffix (*SP* for specs, *PD* for process data sheets, *CL* for calculation sheets, and so forth).

2. Right-click the specification document **101-FV -100-SP**.
3. On the shortcut menu, click **Actions** > **Open Document** to open the control valve specification.
4. View the control valve values and then click  to close the specification.

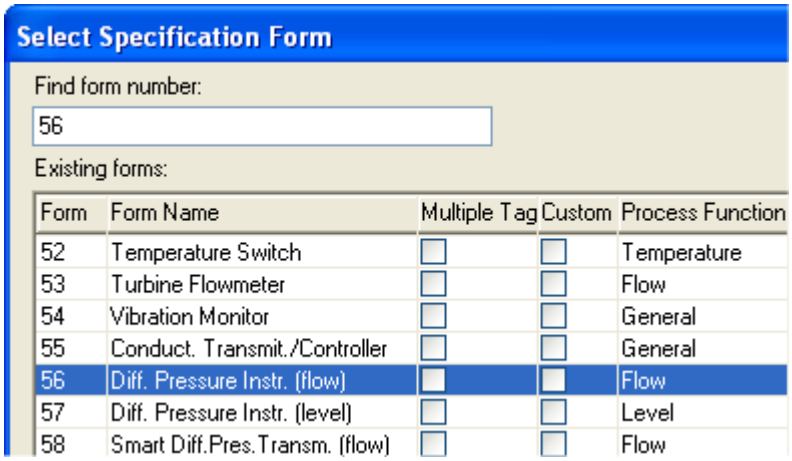
Task 2 — Create a Specification Form Data Template

The specifications you have viewed so far contain specific data relating to each instrument. If you have a large number of instruments of the same type with common data, you can create a form data template and enter the data in this template. You can then use this template for copying this data to instruments of the same type.

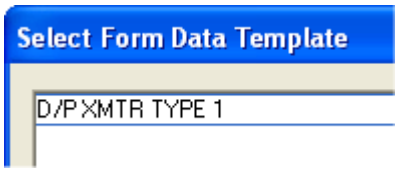
Create a Form Data Template for a Flow Transmitter

1. On the **Specifications Module** window menu bar, click **Edit** > **Form Data Template Editor**.
2. On the **Select Specification Form** dialog box, type **56** in the **Find form number** box.

 The software selects form number **56**, **Diff. Pressure Instr. (flow)** as shown:



3. Click **OK**.
4. On the **Select Form Data Template** dialog box, type **D/P XMTR TYPE 1** as your form data template name.



5. Click **OK** to open the form data template.

6. Enter data as shown.

GENERAL	1	Tag Number			
	2	Service			
	3	Location	P&ID No.	Field	
	4	Function		D/P Flow Indicating Transmitter	
	5	Mounting		2" Pipe	
	6	Area Classification		Class 1, Division 2, Groups C&D	
	7				
	8				
PROCESS CONDITIONS	9	Fluid			
	10	Pressure Max.	Oper.		
	11	Temperature Max.	Oper.	°C	°C
	12	Oper. Spec. Gravity	Oper. Viscosity		cP
	13				
	14				
UNIT	15	Instrument Range Min.	Max.	25 H2O 4°C	
	16	Calibration Range Min.	Max.		
	17	Elevation	Suppression	500% of span	500% of span
	18	Element Type		Piezo Resistive	
	19	Element Material		316ss	
	20	Body Material	Body Rating	316ss	3000 psi
	21	Process Flanges Material		Hastelloy C	
	22	Wetted O-Rings Material		Viton	
	23	Fill Fluid		Silicon Oil	
	24	Bolts	Housing	Cadmium Plated Carbon Steel	Aluminum
	25	Paint		Epoxy	
	26	Process Connection	Electrical Connection	1/2" NPTF	1/2" BSP
	27				
DIAPHRAGM SEAL	28	Process Connection		N/A	
	29	Rating		N/A	
	30	Diaphragm Material		N/A	
	31	Upper Housing Material		N/A	
	32	Lower Housing Material		N/A	
	33	Fill Fluid		N/A	
	34	Capillary Material		N/A	
	35	Capillary Type	Capillary Length	N/A	N/A
	36	Flushing Connection		No	
	37				
	38				
OPTIONS	39	Integral Meter		Yes	
	40	Integral Meter Scale		Yes	
	41	Hydrostatic Testing		Yes	
	42	Cleaning		No	
	43	Calibration		Yes	
	44	Certification		No	
	45				
	46				
	47				
	48				
	49				
PURCHASE	50	Manufacturer		ROSEMOUNT	
	51	Model		3051S1256	
	52	Purchase Order Number			
	53	Price	Item Number		
	54	Serial Number			

Notes: 1. Instrument to be supplied with SS Tag and calibration sheet

TIP You cannot enter data in fields that have a shaded background. These fields are instrument-specific and the values they contain differ from tag to tag.

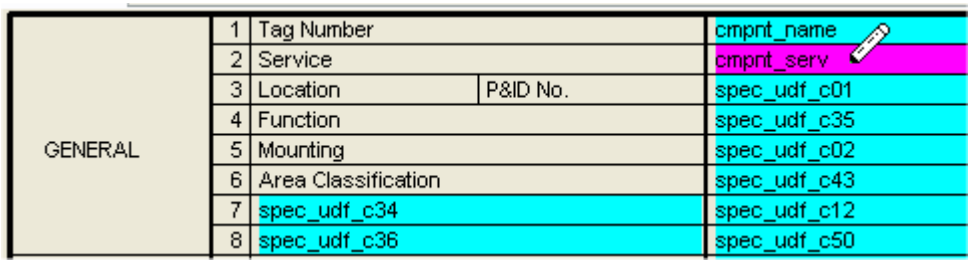
7. When done, on the toolbar, click (**Save**).
8. On the main toolbar, click to close the **Form Data Template Editor**.

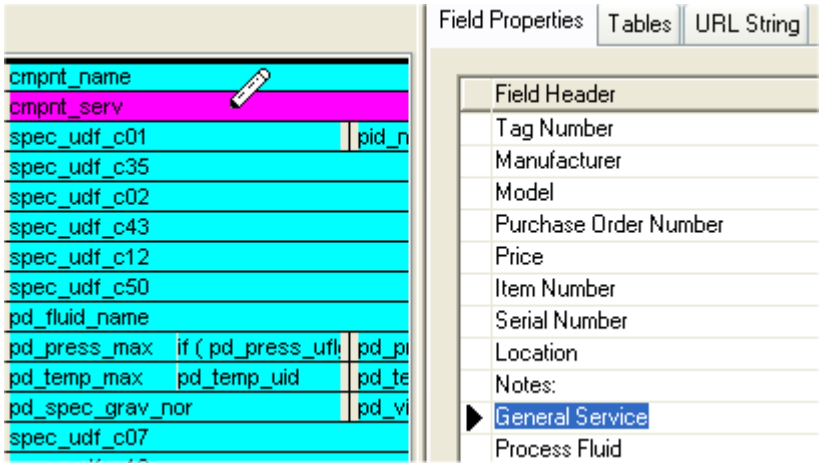
Protect Spec Data Fields from Being Overwritten

Before you copy data from the form data template to specs, you might need to protect certain fields from being overwritten. You perform this procedure in the **Spec Data Dictionary**.

1. On the **Specifications Module** window toolbar, click (**Spec DD**).
2. On the **Select Specification Form** dialog box, type **56** in the **Find form number** box, and then, click **OK**.
3. On the **Spec Data Dictionary** toolbar, click (**Properties**).
4. On the **Field Properties** tab, clear check boxes for the following field headers in the **Template** column:
 - **Manufacturer**
 - **Model**
 - **P&ID No**.
 - **General Service**
 - **General Location**

 TIP You can find field headers in the **Properties** section by double-clicking the field name headers in the working area as shown.





5. Save the changes.
6. Close the **Browser Format** dialog box, and also close the **Spec Data Dictionary**.

Copy Data from a Form Data Template to a Specification

Now you will learn how to automate entry of common data to your specification by using a form data template that you created for the form upon which the flow transmitter spec is based.

1. On the **Specifications Module** window module toolbar, click (**Open Spec**).
2. On the **Open Specification** dialog box, do the following:
 a. Under **Item type,** select **Instrument**.
 b. Under **Item number**, type **101-FT -100**.
 c. Under **Form number**, type **56**.
 d. Click **OK** to open the **Instrument Specification** window for flow transmitter **101-FT -100**.

 TIP The specification was generated when you created the **101-FT -100** tag, according to the instrument type profile settings.
3. In the **Select Fluid State** pop-up window, select **Liquid** and click **OK**.
4. On the **Actions** menu, click **Copy from Template**.
5. On the **Select Form Data Template** dialog box, select the new form data template that you created, **D/P XMTR TYPE 1**.
6. Click **OK** to copy data from the form data template to the specification sheet.

 TIP The values in the protected fields are not overwritten.
7. Save and close the specification.

Task 3 — Creating and Using a Multi-Tag (See-List) Spec

In the previous tasks, you assigned one single tag to one specification sheet. However, there are often cases when one common specification sheet may serve several tags. Such a spec sheet comprises one common data page and an additional SEE LIST page where values can vary for each tag.

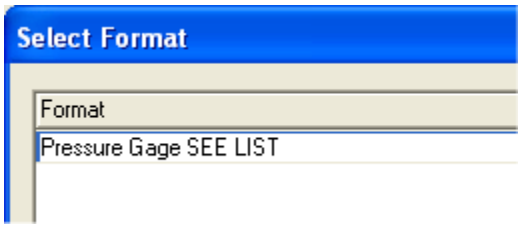
You create a multi-tag spec in two stages:

1. Create a multi-tag spec format on the basis of a particular form and defining fields for the SEE LIST pages of the spec (those fields that can vary from instrument to instrument).
2. Add instruments to the multi-tag spec one by one.

Create a Format for a Multi-Tag Spec

1. In the **Specifications Module** window, on the toolbar, click (**Formats**).
2. In the **Format Editor**, on the toolbar, click (**Open Format**).
3. On the **Select Specification Form** dialog box, select form number **8** and click **OK**.

4. On the **Select Format** dialog box, type a format name as shown.



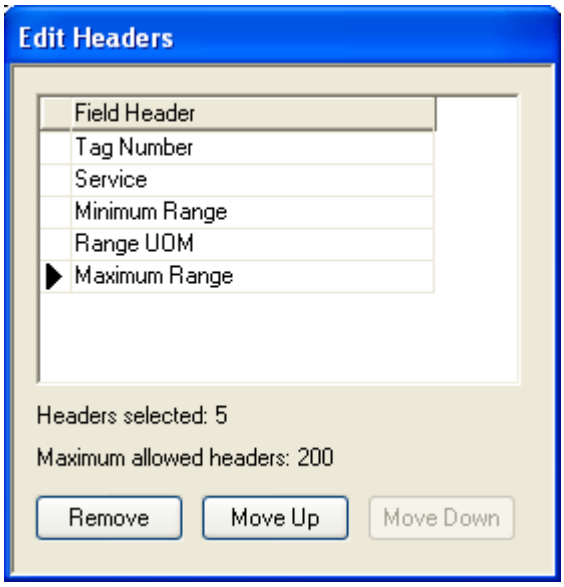
5. Click **OK** to open the **Format Editor** with the **Edit Headers** pop-up window on top of it.

TIPS

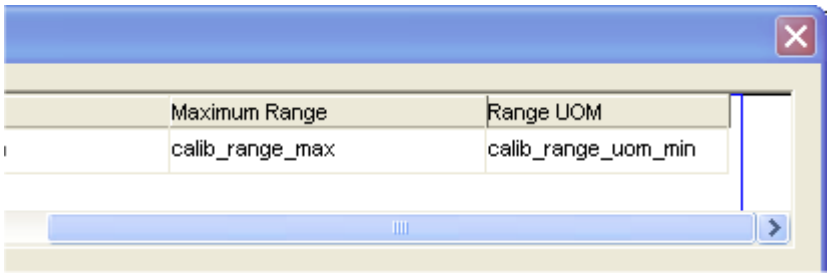
- SmartPlant Instrumentation displays the field headers on the left side of the **Format Editor**, and the field selections on the right side of the window.
- Field headers that you add in the **Edit Headers** pop-up window appear on the SEE LIST page after you generate a multi-tag spec. In the pop-up window, you can modify the sequence of the header names and define the header sequence for the SEE LIST page.

6. Drag the **Edit Headers** pop-up window out of the way so that you can see most of the **Format Editor** window
7. Add the following fields to the **Edit Headers** pop-up window (the **Tag Number** appears in the pop-up window by default):
 a. In the **Format Editor** window, move the cursor over the **cmpnt_serv** (**Service**) field and note that the cursor changes its shape to  .
 b. Double-click in the **field selection** of the **Service** field.
 c. In the **Edit Headers** pop-up window, under **Column Header**, the text **General Service** appears. Edit this text to read **Service**.
 d. Point the cursor to line #13 and double-click in the leftmost field selection, **calib_range_min**. The field changes its color and appears in the **Edit Headers** pop-up window under **Column Header**.
 e. In the **Edit Headers** pop-up window, change the label from **calib_range_min** to **Minimum Range**.
 f. Again in line #13, double-click the second field from the left, **calib_range_uom_min**, and in the **Edit Headers** pop-up window, change the label to **Range UOM**.
 g. Double-click the fourth field from the left, **calib_range_max**, and in the **Edit Headers** pop-up window, change the label to **Maximum Range**. Click **Move Up** to place above **Range UOM**.

h. The headers in the pop-up window should appear as shown. If you make a mistake, you can cancel the field assignment by selecting the selection in the **Edit Headers** pop-up window and clicking **Remove.**



8. On the menu bar, click **Actions** > **Generate Format**.
9. On the **Generate Format** dialog box, drag the **Range UOM** header to the extreme right and modify the header widths by dragging the border between headers to the right or to the left so that the headers appear as shown.



10. Save the format.
11. When done, on the main toolbar, click (**Close**).

You have now successfully created your multi-tag (see-list) spec format, which you can now use for your pressure gauges.

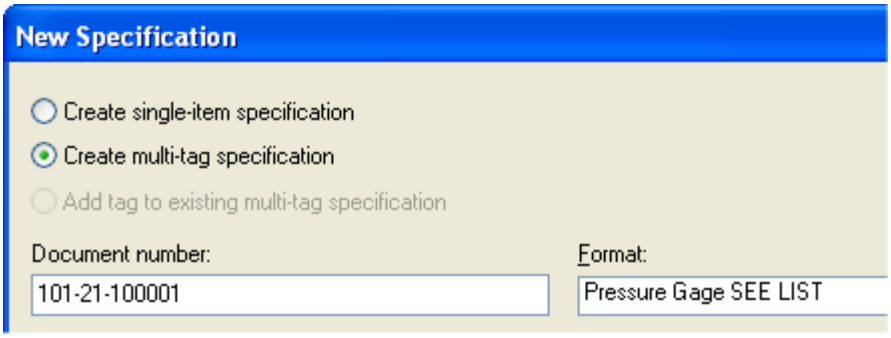
Generate a Multi-Tag Spec

You are now going to generate a multi-tag specification for pressure gauge instruments.

1. On the Specifications Module window toolbar, click (Open Spec).
2. On the Open Specification dialog box, do the following:
 a. Under Item type, select Instrument.
 b. Under Item number, type 101-PI -100.
 c. Under Form number, type 8.
 d. Click OK.
3. On the New Specification dialog box, select Create new multi-tag specification.

4. In the Document number box, type 101-21-100001.
5. From the Format name list, select Pressure Gage SEE LIST.

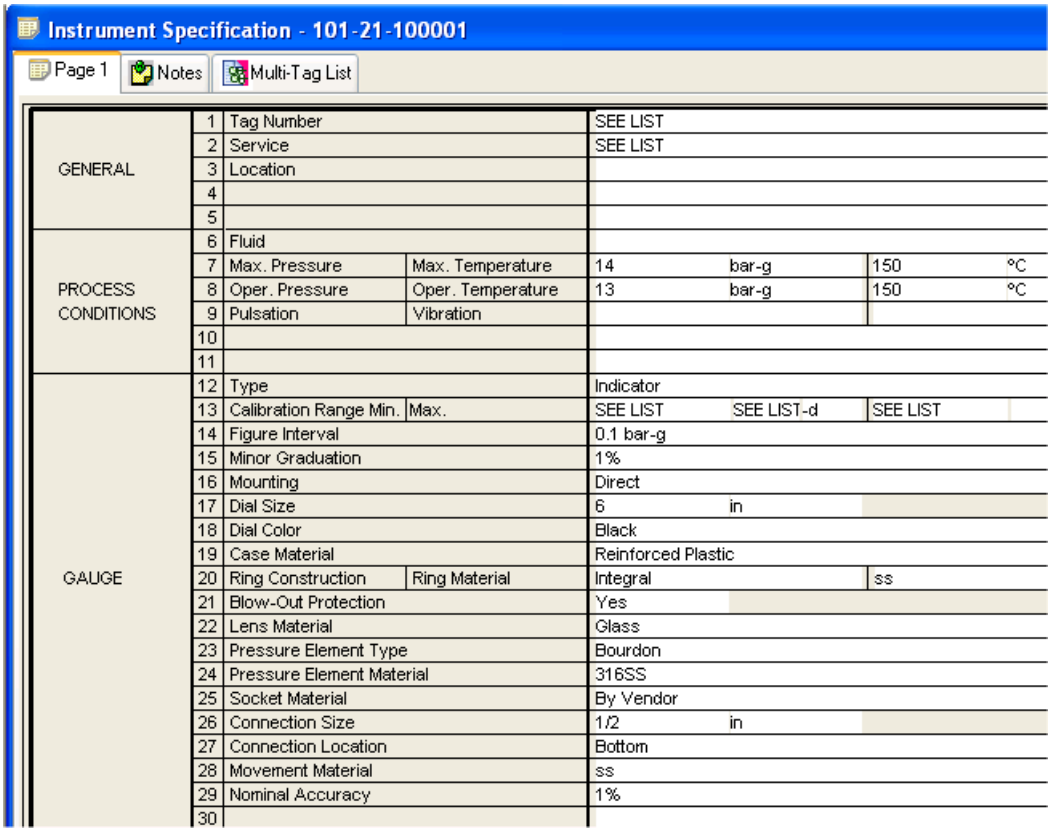
 The dialog box should appear as shown:



6. Click OK.
7. In the Select Fluid State pop-up window, select Liquid and click OK.

 TIP In the data page, the fields that you selected to appear on the following page are marked as SEE LIST.
8. Enter the values in the spec sheet fields as shown.

 TIP For the maximum and operating pressure, in the field area, double-click the field to the left of the vertical field separating line and then select **bar** as the value for **Unit** and **gage** as the value for **Scale**.



GENERAL	1	Tag Number		SEE LIST				
	2	Service		SEE LIST				
	3	Location						
	4							
	5							
PROCESS CONDITIONS	6	Fluid						
	7	Max. Pressure	Max. Temperature	14	bar-g	150	°C	
	8	Oper. Pressure	Oper. Temperature	13	bar-g	150	°C	
	9	Pulsation	Vibration					
	10							
	11							
GAUGE	12	Type		Indicator				
	13	Calibration Range Min.	Max.	SEE LIST	SEE LIST-d	SEE LIST		
	14	Figure Interval		0.1 bar-g				
	15	Minor Graduation		1%				
	16	Mounting		Direct				
	17	Dial Size		6	in			
	18	Dial Color		Black				
	19	Case Material		Reinforced Plastic				
	20	Ring Construction	Ring Material	Integral		ss		
	21	Blow-Out Protection		Yes				
	22	Lens Material		Glass				
	23	Pressure Element Type		Bourdon				
	24	Pressure Element Material		316SS				
	25	Socket Material		By Vendor				
	26	Connection Size		1/2	in			
	27	Connection Location		Bottom				
	28	Movement Material		ss				
	29	Nominal Accuracy		1%				
	30							

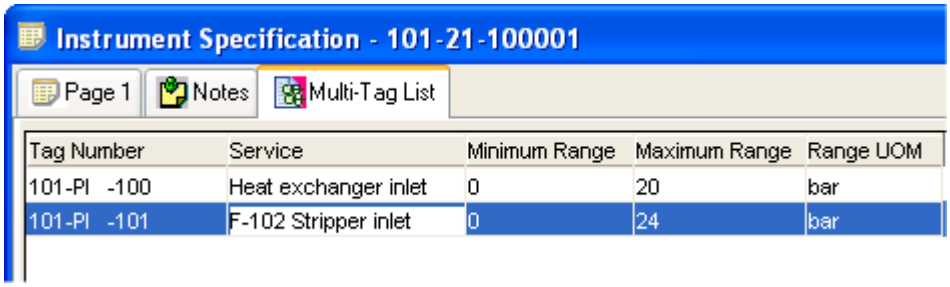
9. On the toolbar, click (Multiple) to display the Multi-Tag List page.

10. Enter the following information:
 a. In the Service column, increase the column width and type Heat exchanger inlet.
 b. In the Minimum Range column, type 0.
 c. In the Maximum Range column, type 20.
 d. In the Range UOM column, select bar.
11. When done, on the Instrument Specification window toolbar, click .

Add an Existing Tag to a Multi-Tag Spec

You are now going to assign an existing tag number to the multi-tag specification that you just created and then you will enter some data.

1. With the **Multi-Tag List** tab displayed, on the toolbar click (**Add Tag**).
2. On the **Add Tag Number** dialog box, type **101-PI- 101,** and then, click **OK**.
3. In the **Select Fluid State** pop-up window, select **Liquid** and click **OK**.
4. On the **Multi-Tag List** tab, enter the specific data and ranges for the additional tag as shown.

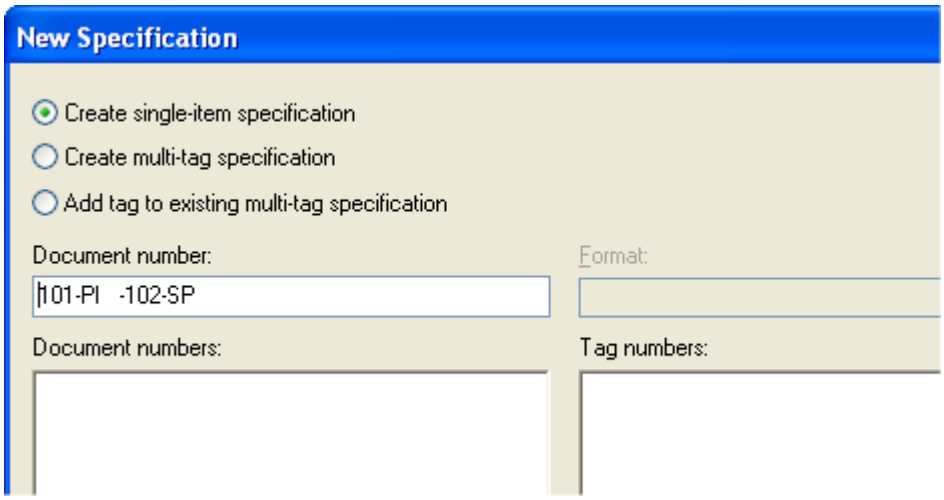


5. On the **Instrument Specification** window toolbar, click (**Save**).
6. On the main toolbar, click (**Close**).

Create a New Tag and Associate it with a Multi-Tag Spec

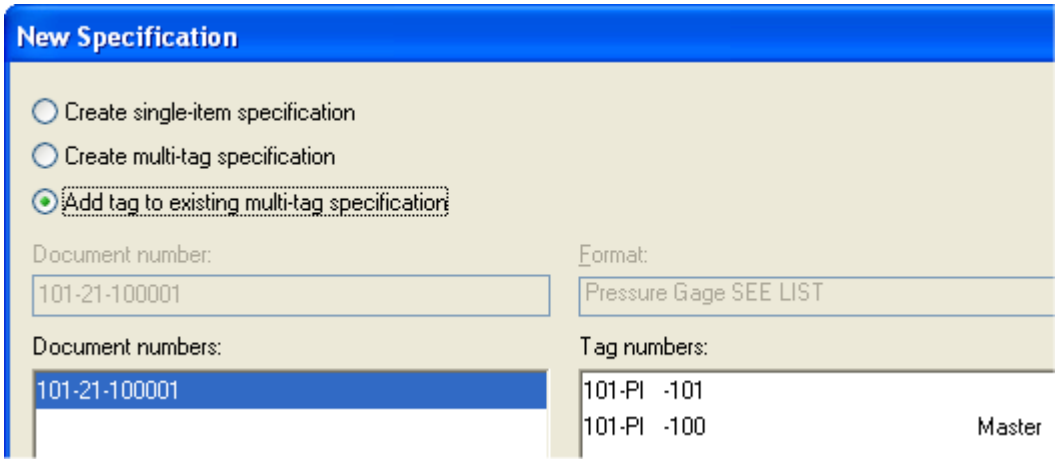
1. On the **Specifications Module** window toolbar, click (**Open Spec**).
2. Do the following to create a new pressure gauge tag:
 a. On the **Open Specificatio**n dialog box, in the **Item number** box, enter **101-PI -102**.
 b. When prompted to create a new tag number, click **Yes**.
 c. On the **Select Instrument Type** dialog box, select **PI PRESSURE GAUGE**, and then, click **OK**.
 d. On the **Loop Name** dialog box, click **Cancel**.
 e. On the **Tag Number Properties** dialog box, click **OK** to return to the **Open Specification** dialog box.
3. On the **Open Specification** dialog box, in the **Form number** box, enter **8**.
4. Click **OK**.

The **New Specification** dialog box should appear as shown:



5. Select **Add tag to existing multi-tag spec**.

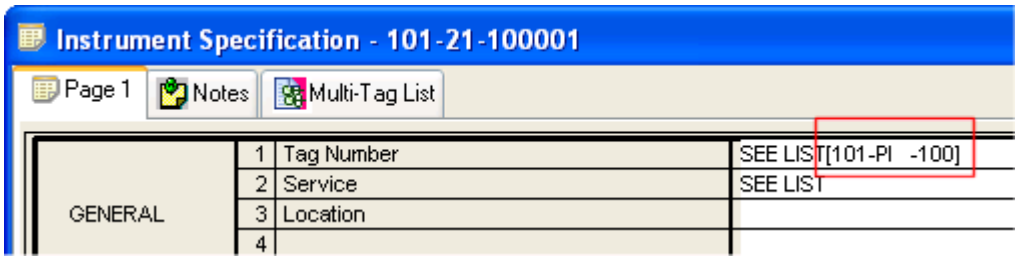
 The **New Specification** dialog box should now display the existing document number and the tags that you added in the spec in the previous procedures:



6. Click **OK**.

 TIPS

 - If the **Select Fluid State** pop-up window displays, select **Liquid** and click **OK**
 - On the common data page, note that the software now displays the master tag.



7. Add some values in the **Multi-Tag List** page, and then save and close the specification.

Task 4 — Customizing Spec Pages

Specification pages are the primary templates for spec forms, on which you base your specs. SmartPlant Instrumentation comes shipped with a number of spec pages that are included in spec forms.

A spec page always contains a number of data fields that correspond to table columns in the database. A spec page also contains field headers that correspond to column headers in the database. You can customize the data fields and field headers to fit specific requirements. You cannot, however, create new pages in SmartPlant Instrumentation.

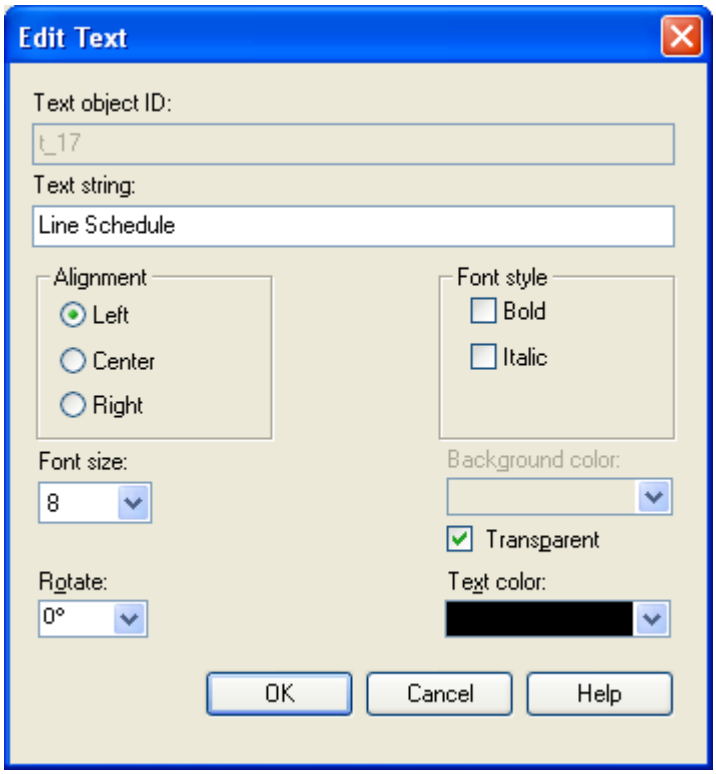
When you assign a form to an item and generate a spec, the software creates the spec pages according to the pages that the source form contains. Changes that you make on the page automatically take effect in all existing specs that include this page. These changes do not take effect in specs saved as files. For example, if you change the thickness of a specific line on a page, and then save the page to the database, the software updates the line display in all specs based on the form containing this page.

Create a Customized Spec Page

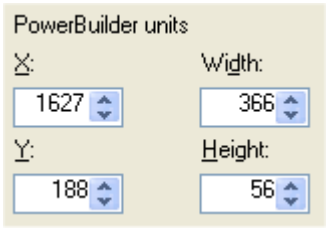
You will now customize a new page called **Modified Flow Element**, change and move some headers and their associated data fields. You will switch the location of two fields: **Line Size** and **Line Schedule**.

1. On the module toolbar, click  to open the **Page Editor**.
2. On the **Page Editor** module toolbar, click  (**Open Page**).
3. On the **Select Specification Page** dialog box, in the **Find page** box, type **Orifice**.
4. In the **Specification Page** column, select the **Orifice Plate** page and click **OK**.
5. On the module toolbar, click  (**Edit**) to open the **Edit Fields and Headers** toolbar on the right.
6. On the specification page, double-click the **Line Size** header to open the **Edit Text** dialog box.
7. In the **Name** box, type **Line Schedule** over the displayed field header.

The dialog box should appear as shown:

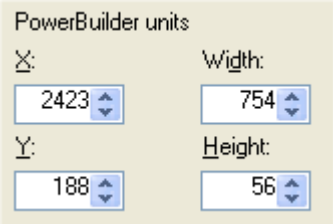


8. Click **OK**.
9. Repeat steps **6** through **8** to swap the text of the **Line Size** and **Line Schedule** headers.
10. Swap the positions of the data fields belonging to **Line Size** and **Line Schedule** as follows:
 a. On the **Edit Fields and Header**s toolbar, click (**View Names**).
 b. Click the **line_size** field.
 c. On the **Edit Fields and Headers** toolbar, click (**Position**) to display the pop-up window showing the position of the **line_size** data field.
 d. Make a note of the field position values.

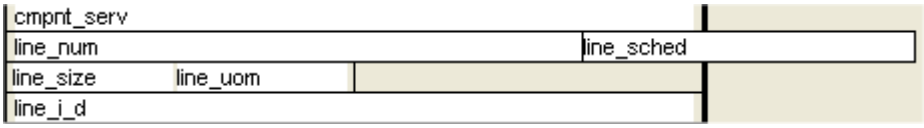


 e. Select the **line_sched** data field and click (**Position**) to display the field position values.

f. Make a note of the X,Y values.



g. Select the **line_sched** data field and drag away from its original position so that the original position becomes vacant.



h. Drag the **line_uom** data field to the right of the **line_size** data field.

i. Select the **line_size** field again and display the **Field or Header Position** pop-up window.

j. In the pop-up window, in the **X** box type **2423**.

k. Select the **line_sched** field.

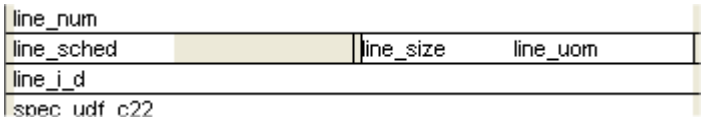
l. In the pop-up window, enter the former **line_size** value for **X** (**1627**). Verify that the value of **Y** is **188**.

m. Select the **line_uom** field.

n. In the pop-up window, enter **1998** as the value for **X**. Verify that the value of **Y** is **188**.

11. When done, click (**Position**) again to close the **Field or Header Position** pop-up window.

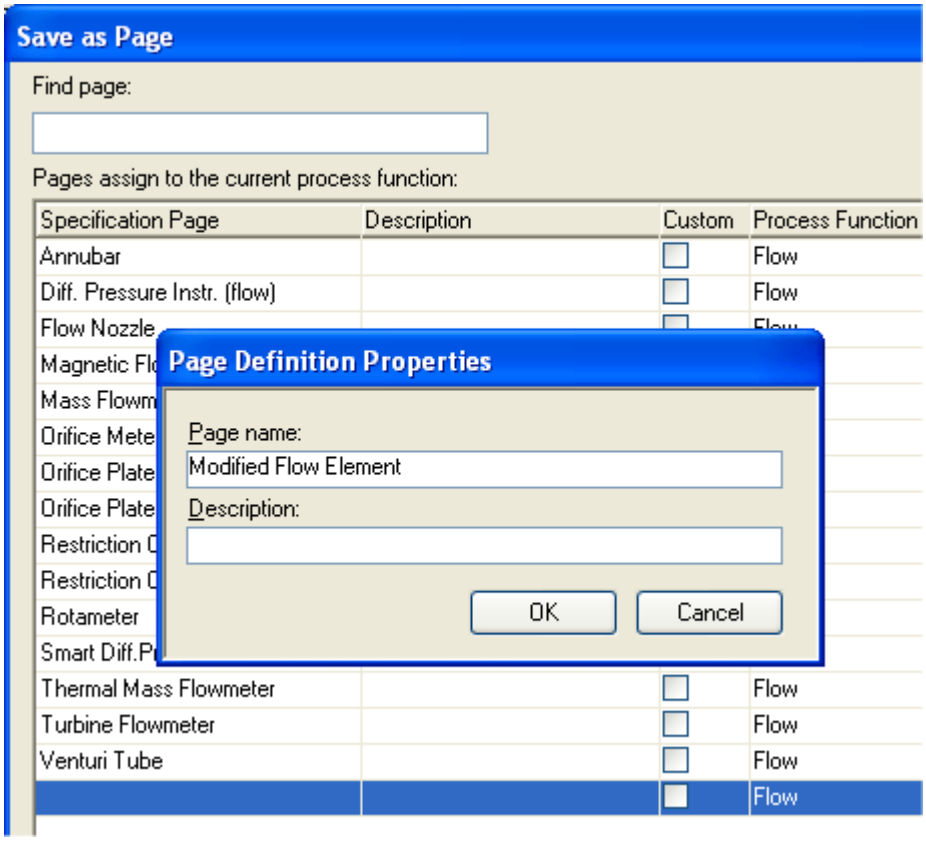
 The modified field positions should appear as shown:



12. On the **Page Editor** toolbar, click (**Edit**) to exit the edit mode and close the **Edit Fields and Headers** side toolbar.

13. On the **Page Editor** toolbar, click (**Save as Page**) on the module toolbar to save the modified page as a new page.

14. On the **Save as Page** dialog box, click **New** to open the **Page Definition Properties** window, and then type the new page name as shown.

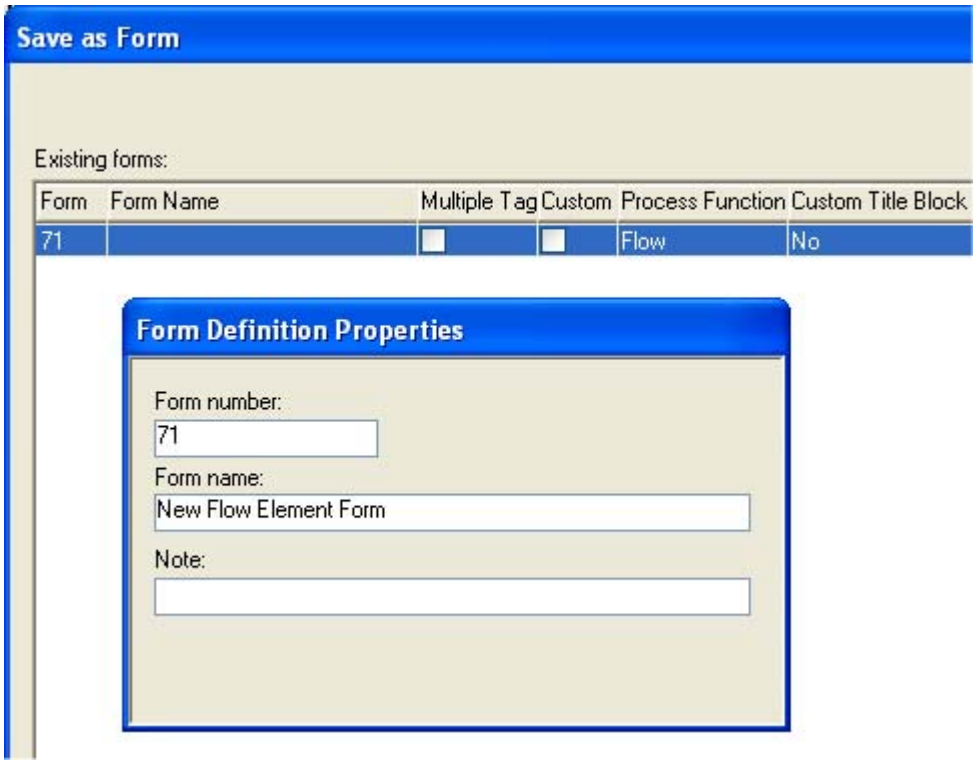


15. Click **OK** to create the new page.
16. Click **OK** to close the **Save as Page** dialog box.
17. Close the **Page Editor**.

Create a Form Containing the Modified Flow Element Page

1. On the **Specifications Module** window toolbar, click (**Form Editor**).
2. On the **Form Editor** module toolbar, click (**New Form**).
3. On the **Select Specification Page** dialog box, select the **Modified Flow Element** page and click **OK**.
4. On the **Form Editor** toolbar, click (**Save as Form**).

5. On the **Form Definition Properties** dialog box, enter information as shown.



TIP In the **Form number** box, the software displays automatically the first available form number.

6. Click **OK** on the **Save as Form** dialog box to save the new form.

Create a Spec for Flow Orifices Based on the Custom Form

1. Switch to the **Specifications Module** window.
2. On the module toolbar, click (**Open Spec**).
3. On the **Open Specification** dialog box, do the following:
 a. Under **Item type**, select **Instrument**.
 b. Under **Item number**, type **101-FE -201**.
 c. Under **Form number**, type **71**.
 d. Click **OK**.
 e. On the **New Specification** dialog box, in the **Document Number** text box, type **101-FE-201-SP**, and then click **OK**.
4. In the **Select Fluid State** pop-up window, select **Liquid** and click **OK** to open the specification and note the new positions of the **Line Schedule** and **Line Size** fields.
5. Save and close the specification.

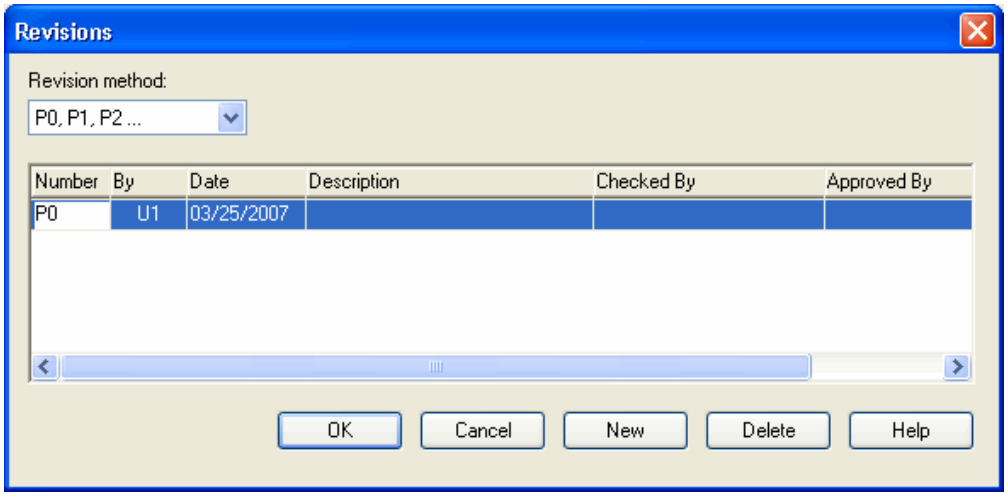
Task 5 — Specification Revisions

During the lifecycle of a plant, you will probably make changes to specifications for any number of reasons; for example, an instrument manufacturer may change some of the characteristics of an instrument, the process conditions may change, or you may just want to add a note explaining the conditions under which the instrument will operate. The revision feature is used to keep track of these changes.

NOTE You can view previous revisions for the specifications that were revised. Prior to this, you must activate the revision archiving option in the Administration module. For details, see *Define Archiving Options for Reports* (on page 36). With this option, you can save the specifications with associated data to the database or to an external file.

Add Revisions to Specification Sheets

1. Open the specification for tag **101-FE -201**.
2. Click to open the **Revisions** dialog box.
3. Click New to add a revision to the specification sheet.



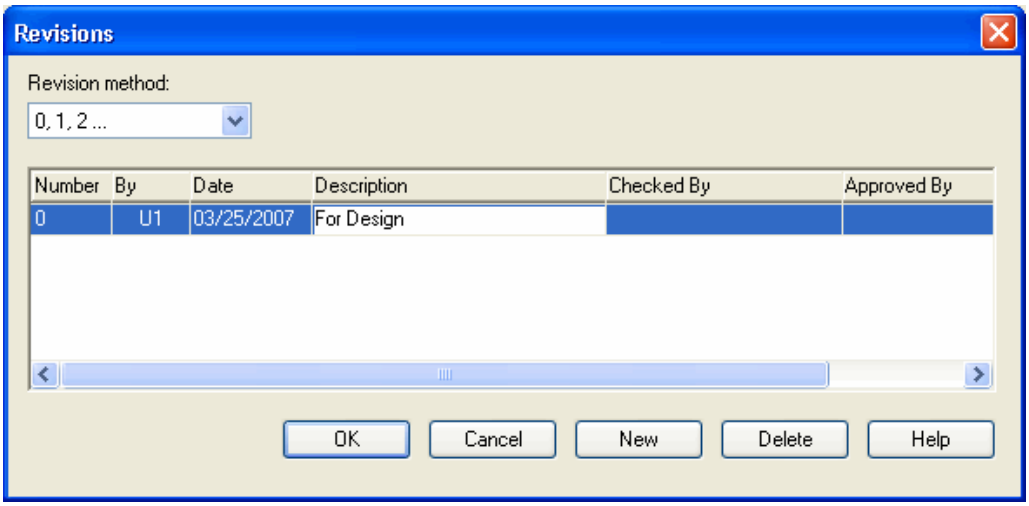
4. Click **OK** to save the revision and close the dialog box.
5. Close the specification sheet.
6. When prompted to save data, click **Yes**.
7. Open the specification sheet for tag **101-FE -100**.

 TIP You may see a message indicating that process data has been changed. Select **Do not remind me again** and click **OK** to continue.

8. Click to open the **Revisions** dialog box.
9. Enter revision data as follows:
 a. Select **Revision method 0, 1, 2…** and click **New**.

 TIP When you first select a revision numbering method, several options are available to you, including preliminary revisions (designated by P0, P1, P2…). Once you select one of the other revision methods, you will not be able to return to the preliminary revision method and this option will be disabled.

b. In the **Description** field, type **For Design**.



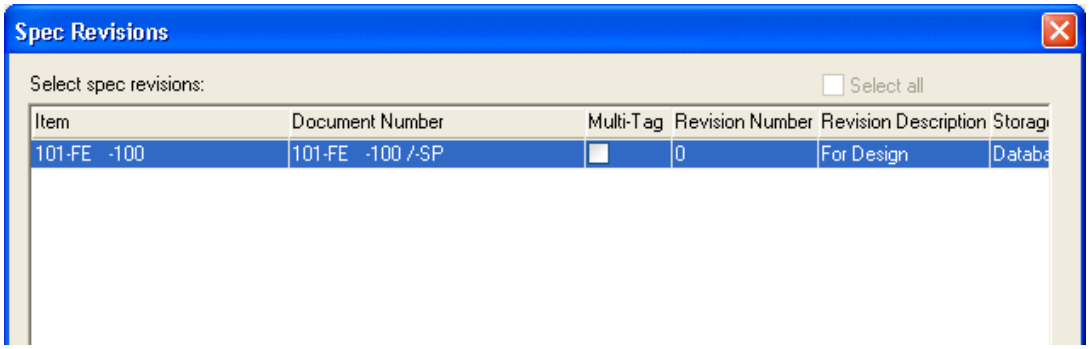
10. When completed, click **OK** to return to the specification sheet.
11. Click (**Save**).
12. Make the changes to the following tag number properties in the specification sheet:

Property	**Initial Value**	**Changed Value**
Service	Feed from V8	Feed from V8-A
Fluid	Lean Feed	Vapor Feed
State	Liquid	Gas/Vapor

13. Click (**Save**).
14. When prompted to re-archive the specification, click **No**.

Generate a Comparison Report Between the Current Spec and an Archived Revision

1. With the specification sheet for tag **101-FE -100** open, click **Actions** > **Compare with Revision**.
2. On the **Spec Revisions** dialog box, select the revision you created (Revision Number **0**).



3. Click **OK**, and then, when prompted to preview the report, click **Yes**.

The Specification Comparison Report displays the values in the current spec sheet and in the revision:

Specification Comparison Report

Page 1 of 1

Column Name	Current Value	Revision Value
Item Number :	101-FE -100	Rev_no : 0 Rev_desc: For Design
Notes: (spec_note)	1. Orifice flange assembly to be provided by piping.	See notes
General Service (cmpnt_serv)	Feed from V8-A	Feed from V8
Process Fluid (pd_fluid_name)	Vapor Feed	Lean Feed
Process State (pd_fluid_phase)	Gas/Vapor	Liquid

4. Close the comparison report and view the marked properties in the spec sheet.

1	Tag Number		101-FE -100			
2	Service		Feed from V8-A			
3	Line Number		4"-P-1501-11H			
4	Line Size	Line Sched.	4	in	80	
5	Line I.D.		3.826			
6						
7	Fluid	State	Vapor Feed		Gas/Vapor	
8	Flow Max.	Flow Min.	32	Am³/h	25	Am³/h

5. Click **Actions** > **Compare with Revision** again to hide the comparison display colors in the spec sheet.

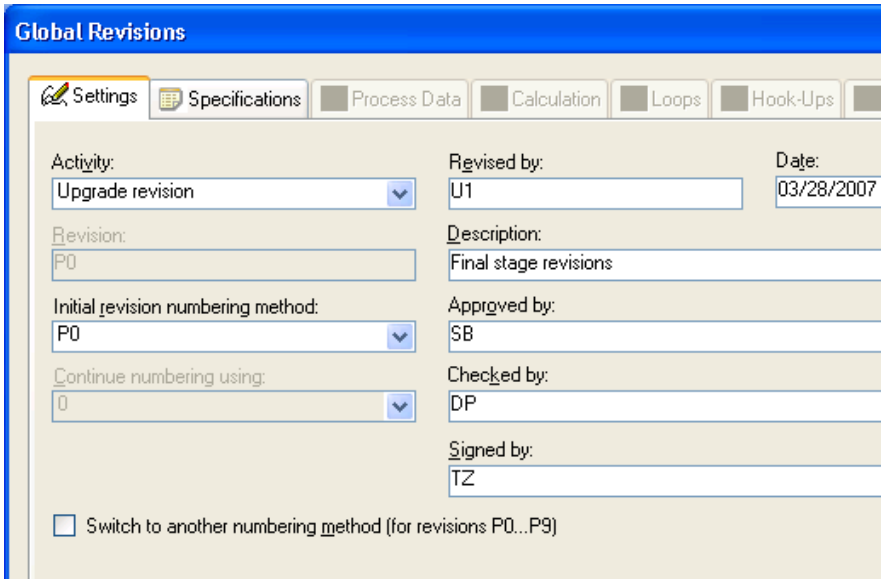
Make Global Revisions for Specifications

Sometimes, when you make a change in the specification sheet of a particular instrument, you will want to create a formal revision indicating when the change was made, who made the change, and who approved it. In other cases, the specifications of a whole class of instruments may have changed. When this happens, you will probably want to revise a complete set of specification sheets. For this purpose, the global revisions option is available.

1. With the **Specifications Module** window displayed, on the **Tools** menu, click **Global Revisions**.
2. On the **Global Revisions** dialog box, click the **Settings** tab, and from the **Activity** list, select **Upgrade revision**.
3. Enter the required revision numbering criteria to be used as the default in all the newly created revisions as follows:
 a. From the **Initial revision numbering method** list, select method **P0** (preliminary revision numbering). This method will be used for specification sheets that have no previous revisions.
 b. Select the options to change the revision numbering type for all selected specification sheets that have **preliminary revisions** (revision numbering P1, P2, and so forth). To do this, select the **Switch to another numbering method** check box, and from the **Continue numbering using** list, select method **A** (alphabetic revision numbering).

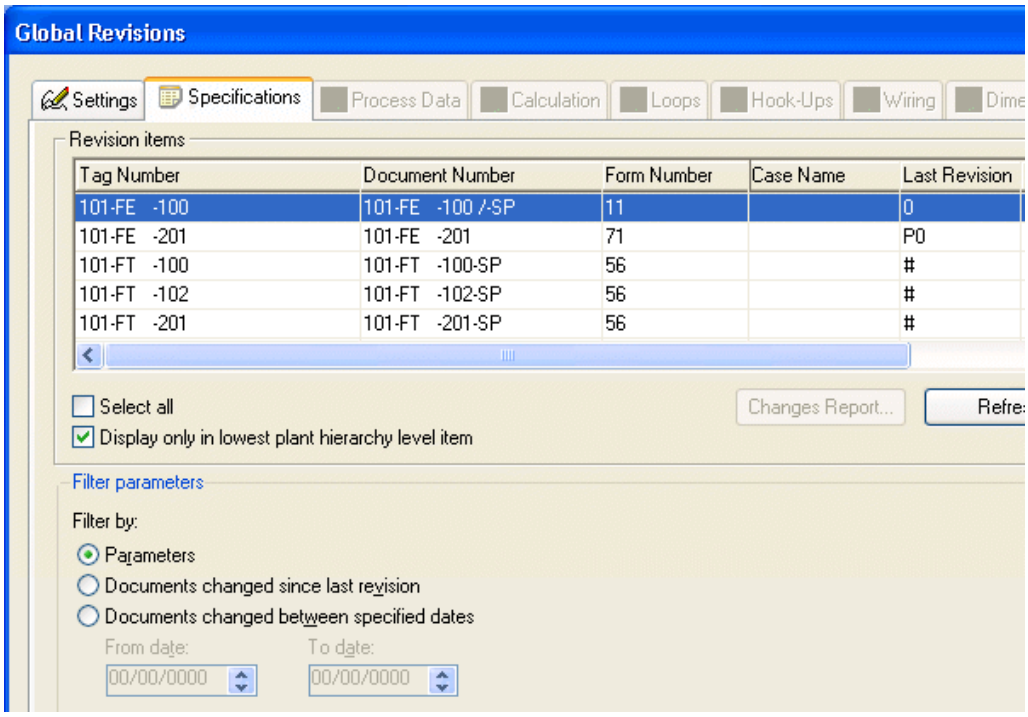
TIP Revision numbering for other specification sheets will be based on the existing numbering method, and will be incremented from the previous revision.

4. Complete the information for the other fields on the **Settings** tab (**Revised by**, **Revision Date**, and so forth) as desired, so that it appears similar to that shown.



5. Click the **Specifications** tab.
6. Select the check box **Display only in lowest plant hierarchy level item**.

 The software filters the list of tag specification sheets to reflect the selection:



7. Select the following specification sheets (while holding down **Ctrl**):
 - **101-FE -100**
 - **101-FE -201**
 - **101-FT -100**

8. Click **Apply**.

 The software applies revision upgrades to the selected specification sheets. A progress bar indicates the percentage completion of the process.

9. Click **Refresh** to update the data.

 Note the following revision number changes for the selected specification sheets:

 - **101-FE -100** (last revision number **0**; new revision number **1**).
 - **101-FE -201** (last revision number **P0**; new revision number **0**).
 - **101-FT -100** (no last revision; new revision number **0**).

SECTION 9

Managing Documents

SmartPlant instrumentation enables you to gather documents and maintain them together as a document packages, via the Document Binder module. Each collection of documents is known as a binder package. There are two types of binder packages that you can create:

- Specification Binder packages, which can contain specification sheets only.
- General Document Binder packages, which can contain various types of documents, including specification sheets.

You can group documents according to any criteria you require, for example, instruments associated with a particular vendor, location, or plant design phase.

NOTE A given specification sheet may only be assigned to **one** Specification Binder package or to multiple General Document Binder packages, but not to both.

In this part of the tutorial, you will:

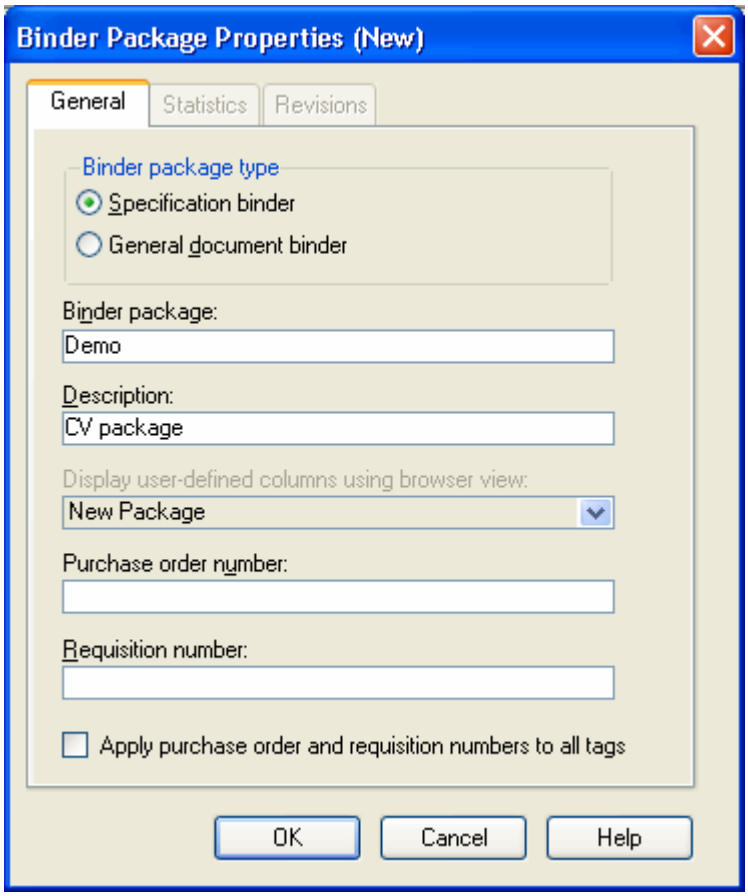
- Create a Specification Binder package for control valves that you defined previously.
- Create a form note template, and add a form note based on that template.
- Add a General Note to the Specification Binder package.
- Modify specification sheets in the Specification Binder package and add revisions in the Document Binder module.
- Print documents for the Specification Binder package.
- Create a General Document Binder package for calibration reports.

Task 1 — Creating a Specification Binder Package

Create a Specification Binder Package

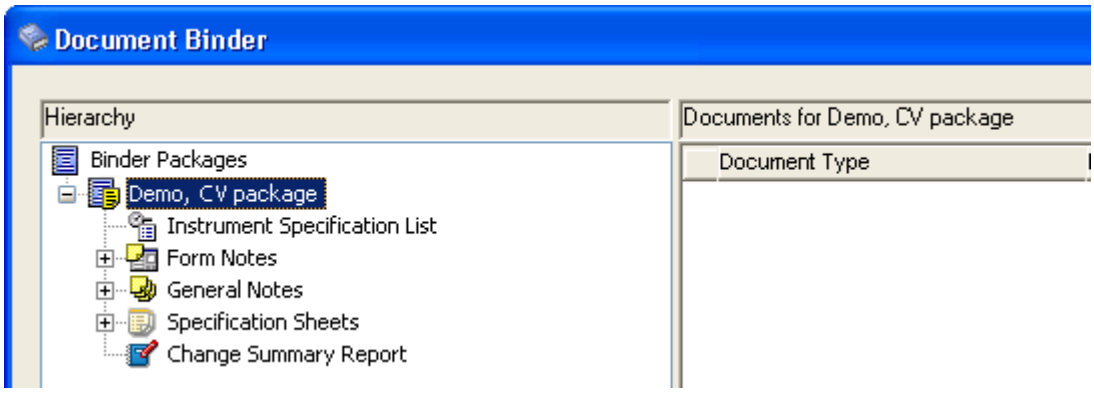
1. To open the Document Binder module, do one of the following:
 - On the main toolbar, click .
 - On the main menu bar, click **Modules** > **Document Binder**.
2. Select the **Binder Packages** icon  at the top of the tree view.
3. Right-click the icon and on the shortcut menu, click **New Binder Package**.
4. On the **Binder Package Properties** dialog box, under **Binder package type**, select **Specification binder**.

5. Enter a binder package name and description as shown.



6. Click **OK**.

 The software automatically creates a document hierarchy for the Specification Binder package with the right pane displaying the column headers for the documents included in the binder package:



 Currently, the columns are empty because you have not assigned any documents to the binder package. In the next part of this task, you will assign specification sheets for some of the instrument tags you already created. Later, you will add other documents.

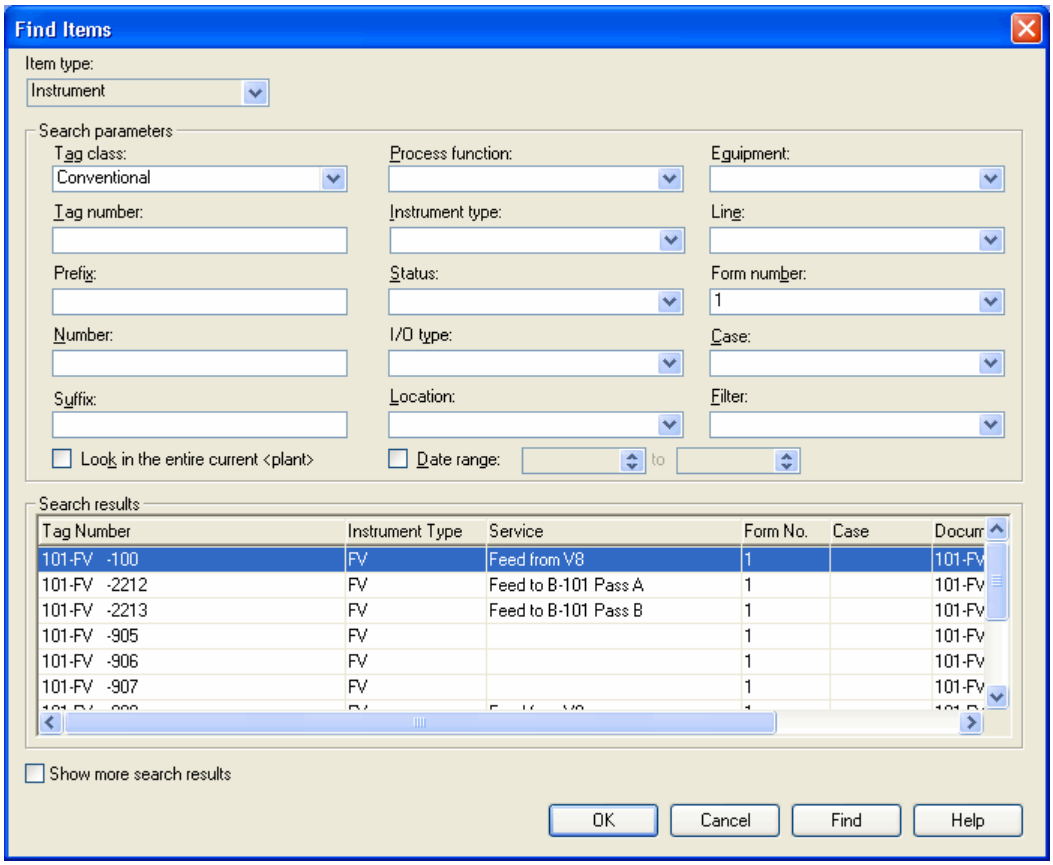
Assign Specification Sheets to the Binder Package

You are now going to assign a number of specification sheets to the instrument tags you created based on Form No. 1 (control valves).

1. If necessary, expand the folder **Demo**, **CV package**.
2. Right-click the **Specification Sheets** folder.
3. On the shortcut menu, click **Assign Specification**.

 The **Find Items** dialog box opens, where you can search for the desired tags.
4. In the **Form Number** list, select **1 (Control Valve)** and click **Find**.

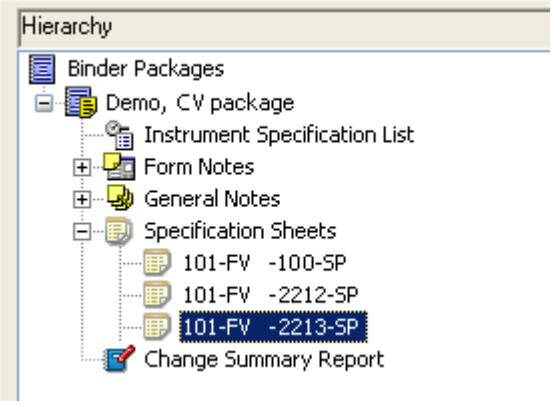
 The dialog box should appear as shown:



TIP Depending on which tags you created in the Instrument Index module, the list of tags that appear on your screen may differ from those shown above.

5. From the list, select the following tags for inclusion in the binder package: **101-FV -100**, **101-FV -2212**, and **101-FV -2213**.
6. Click **OK**.

The list of tags appears under the **Specification Sheets** folder for the binder package:



7. In the **Hierarchy** tree view, click the **Instrument Specification List** and **Specification Sheets** folders and view the information displayed in the right pane of the window.

NOTE The specification list is a cover sheet with instrument tag data for all the specification sheets included in the binder package. You can also configure the layout and the fields to be displayed using the **Preferences** window for the Document Binder module (for further details, see *SmartPlant Instrumentation User's Guide*, *Document Binder*).

Task 2 — Creating Form Notes and General Notes

You can add two types of notes to a binder package. The following examples include both:

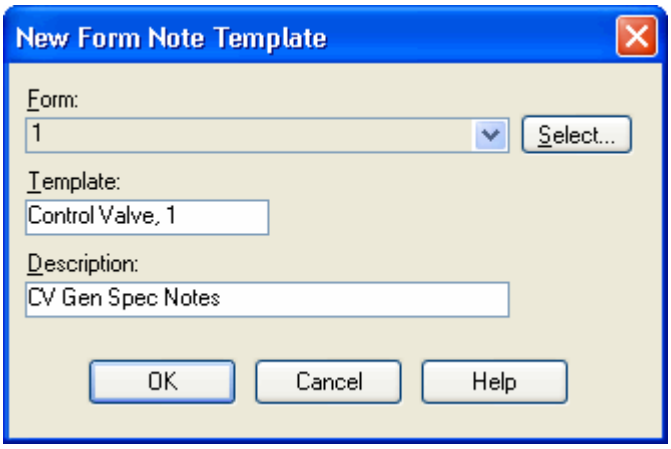
- **Form Notes** — These are user notes that you assign for specification sheets based on a particular form. While there is no limit to the number of notes that you can add per form, at least one specification sheet associated with that form must exist in the binder package.
- **General Notes** — General notes provide means of adding user comments that are associated with the whole binder package.

Create a Form Note Template

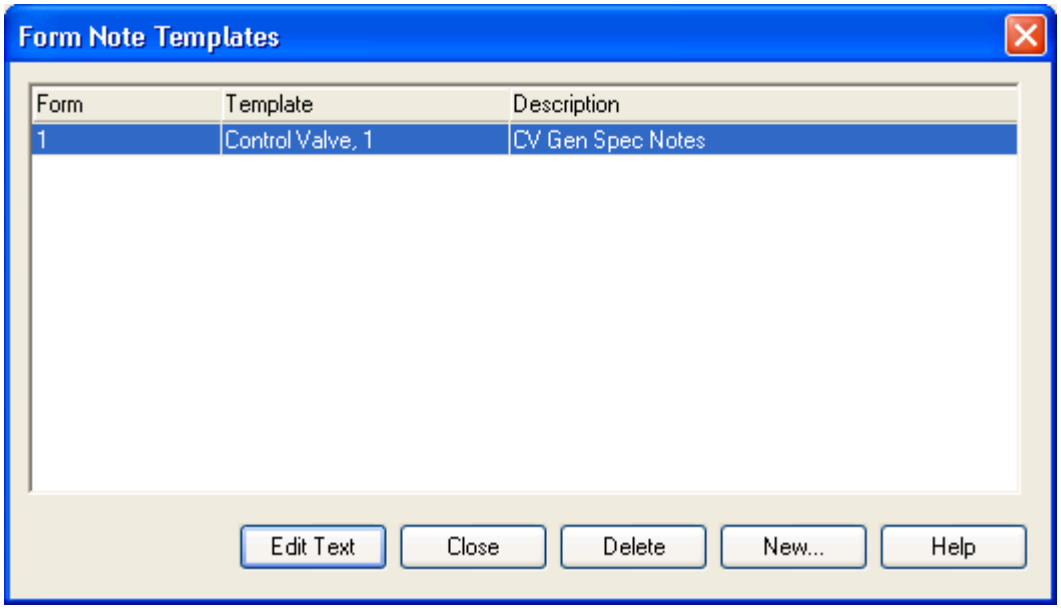
A form note template provides a convenient means of creating standard text that can be used in a large number of form notes, each of which can then be edited specifically as required.

1. Click **Actions** > **Form Note Templates**.
2. Click **New** to open the **New Form Note Template** dialog box.
3. Enter the following template definitions:
 a. From the **Form** list, select **Form No. 1 (Control Valve)**.
 The text **Control Valve, 1** (the form name) appears in the **Template** field.

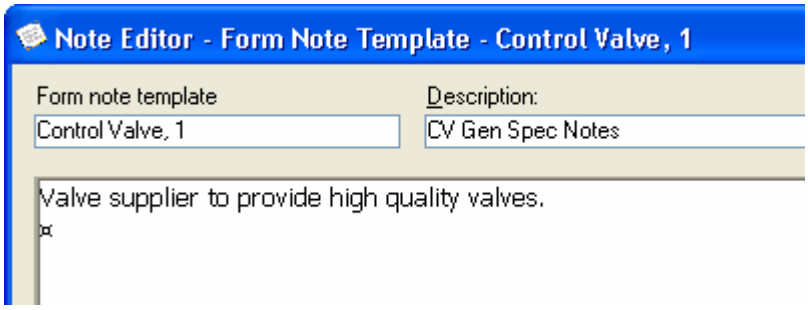
b. Under Description, type text as shown.



4. Click **OK**.
 You are returned to the **Form Note Templates** dialog box and the template appears in the list:



5. Select the row for the template you just created and click **Edit Text**.
 The **Form Note Templates** dialog box closes and the **Note Editor** window opens.
6. In the note pane, type text as shown.



7. On the module toolbar, click  to save the note you just typed in the template.

8. Click  to close the note editor and return to the **Document Binder** window.

Create a Form Note from the Template

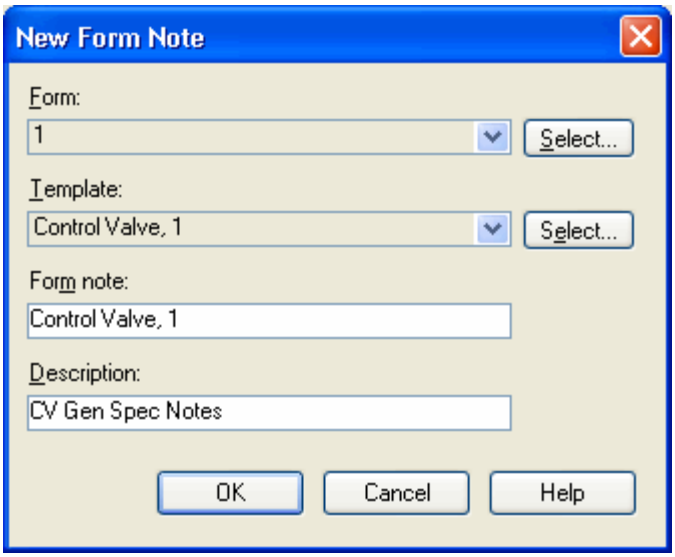
You will now add a form note for the control valves that you have included in the binder package. You will use the template you just created as a basis for the note, which you will edit by adding more text.

1. Right-click the **Form Notes** folder.
2. On the shortcut menu, click **New Form Note**.
3. On the **New Form Note** dialog box, enter a name and description for the form note as follows:
 a. From the **Form** list, select **1 (Control Valve)**.

 TIP In this case, because **all** the specification sheets that you assigned to the binder package use only form number 1, this is the only form that appears in the list.

 b. From the **Template** list, select **Control Valve, 1** (the template you created for the selected form).

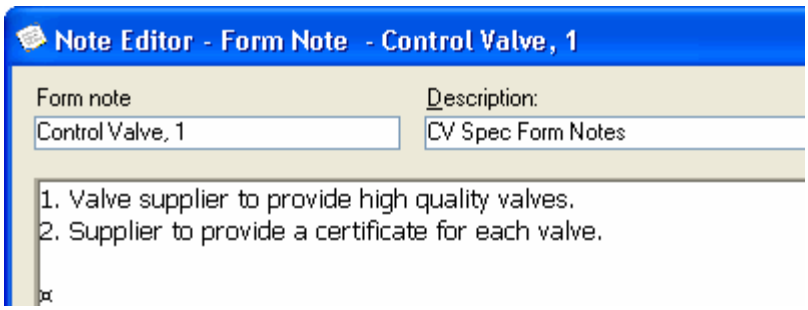
 The dialog box should appear as shown, with the **Name** and **Description** fields automatically filled with the template name and description:



4. Click **OK**.

 The **Note Editor** window opens. The note text that was entered for the template appears by default in the note field.

5. Make changes as shown to the note description and the note text itself as shown in the example.



6. When done, click .
7. Click to close the note editor and return to the main window.

Task 3 — Editing and Revising Specification Sheets from the Binder Package

Using the Document Binder module, you can open specification sheets directly for editing, perform a collective update for the documents in a binder package and assign the same revision number to all.

Edit Specification Sheets

In this part of the task, you will edit two of the specification sheets that you assigned to the binder package.

1. In the **Hierarchy** tree view, click the **Specification Sheets** folder.
2. In the right pane, double-click tag number 101-FV -100 to open its specification sheet in the Specifications module.

 TIP You may see a message indicating that process data has been changed. Select **Do not remind me again** and click **OK** to continue.
3. Scroll the sheet to line #18 (**Flow Rate**) and in the **@Norm. Flow** field, type **29** and ensure that the selected units are **Am^3/h**.
4. On the module toolbar, click to save your changes.
5. On the main toolbar, click to close the specification sheet and return to the Document Binder module.
6. Select the **Specification Sheets** folder and view the information displayed in the right pane.

 The **Changed** column in the right pane displays **Yes** for specification sheets that have not been assigned to a binder package before. It also displays **Yes** when a specification sheet has been modified prior to creating any revision. This indicates that the specification sheet revision number will be incremented the next time you create a revision for the binder package.

Revise a Binder Package for the First Time

A feature of binder packages is that you perform revisions for the binder package as a whole, rather than for the individual specification sheets. For this reason, once you assign a specification sheet to a binder package, you may only perform revisions from the Document Binder module. You are now going to issue a new revision for the binder package.

1. Right-click the binder package folder **Demo**, **CV package**.
2. On the shortcut menu, click **Revisions**.
3. On the **Revisions** dialog box, select revision method **P0, P1, P2…** and click **New**.
4. In the **Description** field, type **Package Rev. 1**.
5. Click **OK**.

 The software compresses and saves the data to the database.
6. Click **Actions** > **Refresh**, and select the **Specification Sheets** folder.
7. In the right pane, verify that in the **Changed** column, the value for each of the specification sheets is **No**. This indicates that the binder package was revised and that no further changes

were made to the specification sheets since that revision (for further details, see the *SmartPlant Instrumentation User's Guide*, *Document Binder*, *Notification of Specification Sheet Changes*). Also, note that all the specification sheets in the binder package have revision number P0.

TIP The first time you perform a revision on a binder package, the software increments the revision number for all the specification sheets. On subsequent revisions, the software only increments revision numbers for which the Changed column value is Yes.

8. To view the revisions that were added to a particular instrument specification sheet, open the specification sheet for that instrument in the Specifications module and click .

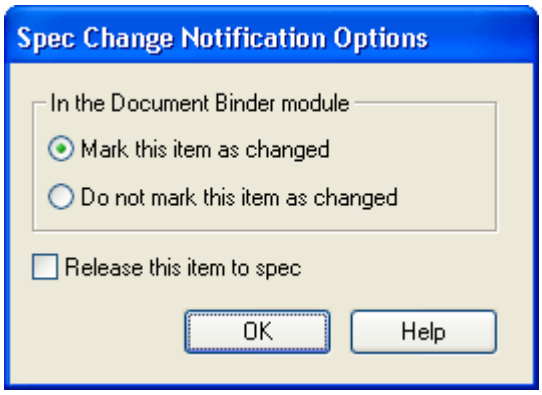
Make Subsequent Changes and Revisions for a Binder Package

You will now modify the same two specification sheets as in the previous objective and then revise the binder package.

NOTE From now on, you can view any performed changes in the Change Summary Report.

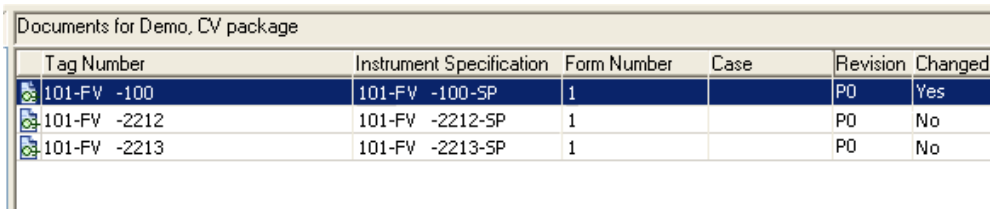
1. Edit the specification sheet for tag number **101-FV -100** once more (making the changes in the same field that you previously changed) and click .

 The following dialog box appears:



2. Select **Mark this item as changed** to set the value of the **Changed** column in the Document Binder module to **Yes** for the specification sheet, indicating that the software will increment its revision number the next time you revise the binder package.
3. On the main toolbar, click to close the specification sheet and return to the Document Binder module.
4. Select the **Specification Sheets** folder and view the values in the **Changed** column in the right pane.

 The display should appear as shown:



Tag Number	Instrument Specification	Form Number	Case	Revision	Changed
101-FV -100	101-FV -100-SP	1		P0	Yes
101-FV -2212	101-FV -2212-SP	1		P0	No
101-FV -2213	101-FV -2213-SP	1		P0	No

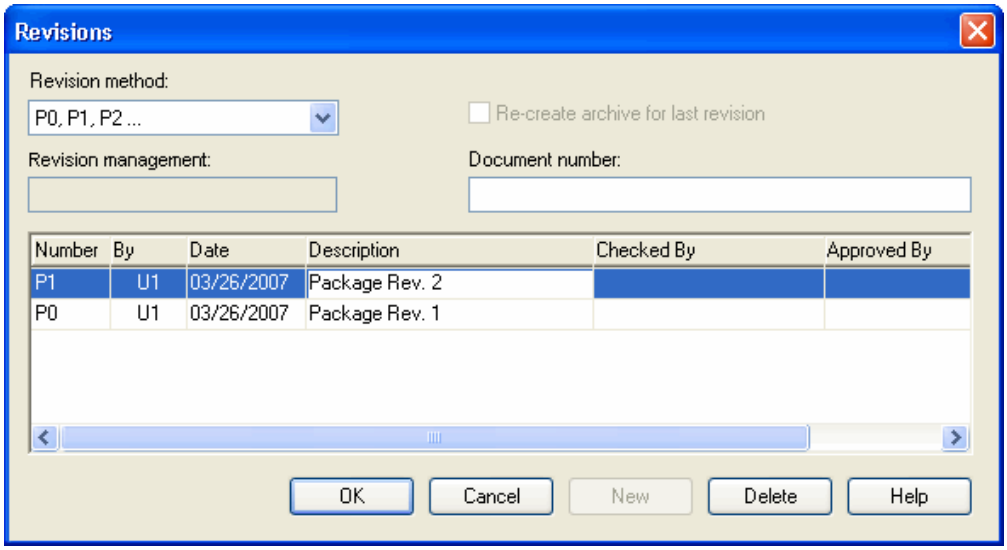
 Note that the value in the **Changed** column for the modified specification sheet is now **Yes**.
5. Double-click the **Change Summary Report** folder.

TIP The **Change Summary Report** compares between a specification sheet and the sheet of the last saved binder package revision. The comparison is performed between values in the specification sheet fields that were modified.

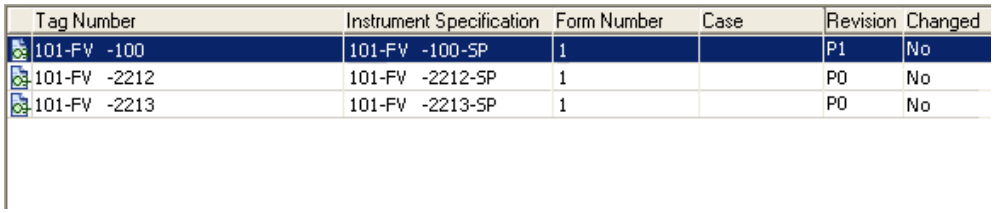
6. Close the **Change Summary Report**.

 Next you will revise the binder package in order to increment the revision numbers of the specification sheets whose data was changed.

7. Create a new revision in the Document Binder module for your binder package.



Note that the revision number in the **Revision** column has changed only for the modified specification sheet, and that all the values in the **Changed** column have been reset to **No**:



8. Double-click the **Change Summary Report** folder.

 The Change Summary Report is empty because no changes were made to the specification sheets since the last binder package revision.

9. Close the **Change Summary Report**.

Task 4 — Printing Documents for a Specific Revision

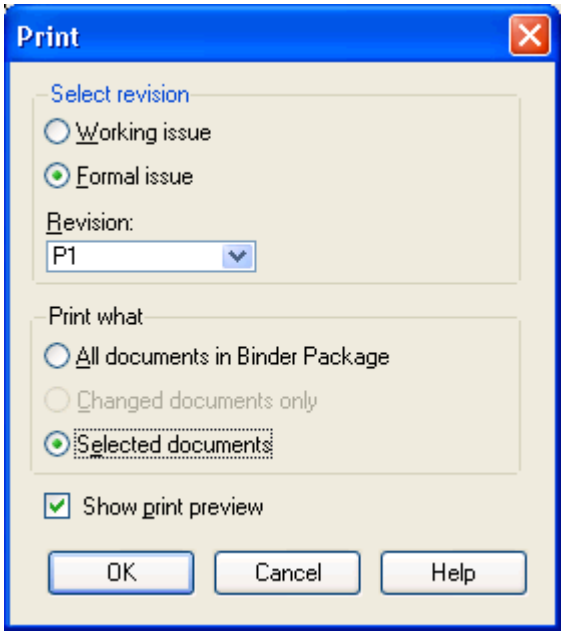
You have already viewed the Change Summary Report for a working issue of the binder package. In this task, you will select a particular revision and select several of the documents associated with that revision to print. Your selection will include a Change Summary Report showing details of the changes that you made to the specification sheets since the last binder package revision.

Print Documents for a Particular Revision

1. Right-click the binder package folder **Demo, CV package**.
2. On the shortcut menu, click **Print**.
3. On the **Print** dialog box, make the following selections:
 a. In the **Select revision group** box, select **Formal issue**.
 b. From the **Revision** list, select **P1**.

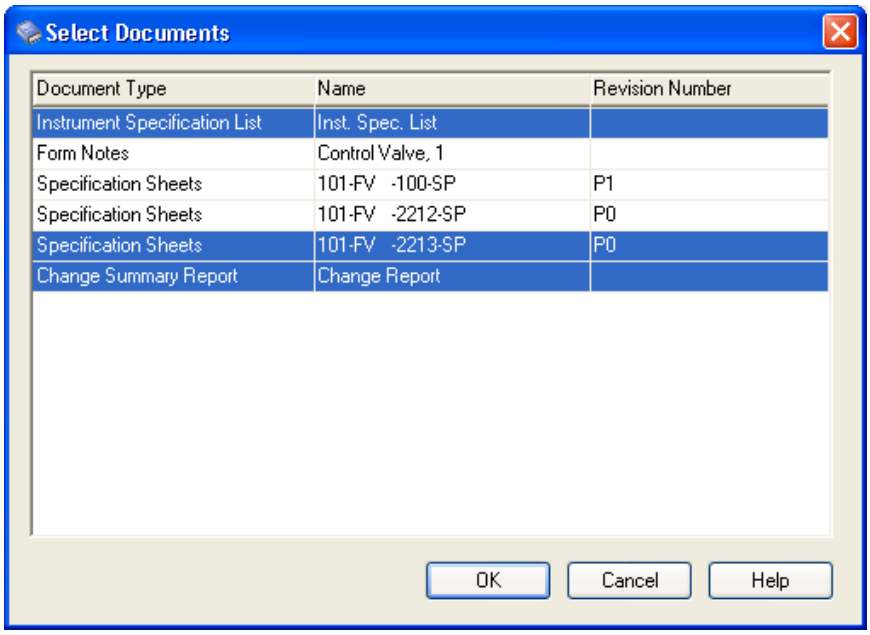
 This will enable you to display a Change Summary Report for changes that occurred between Revisions **P0** and **P1**.
 c. Under **Print what**, select **Selected documents** to be able to select which of the documents in the binder package you want to print.

 The dialog box should now appear as shown:

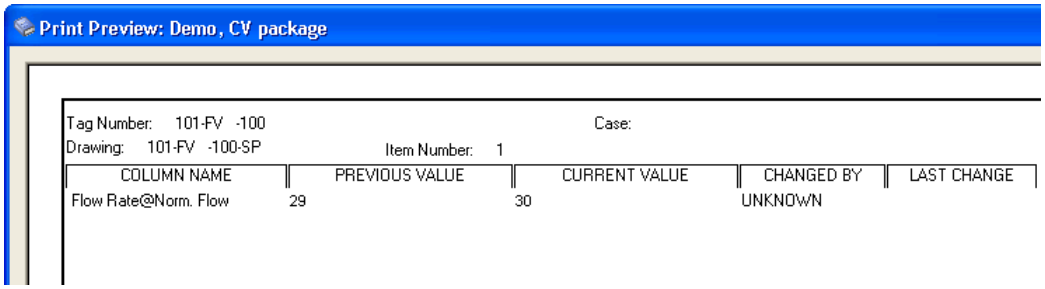


 d. Click **OK**.

4. On the **Select Documents** dialog box, select the documents as shown and click **OK**.



5. In the **Print Preview** window, click  until the Change Summary Report appears (your values may differ from the ones shown according to the changes that you made).

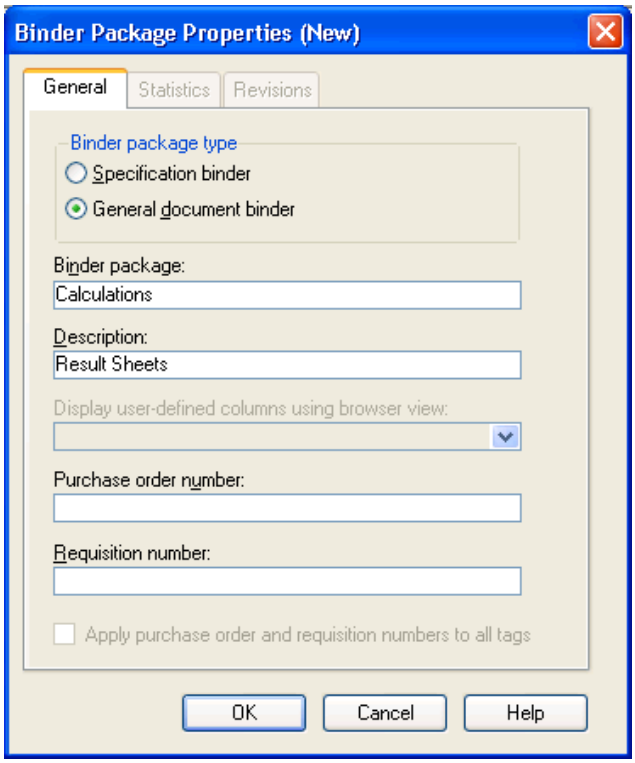


Task 5 — Creating a General Document Binder Package

In this task, you are going to create a new package of the type General Document Binder. In this package, you will add some calculation reports and associate an external document. You can then add a revision to the General Document Binder package.

Create a General Document Binder Package

1. Select the **Binder Packages** icon at the top of the tree view.
2. Right-click the icon and on the shortcut menu, click **New Binder Package**.
3. Under **Binder package type**, select **General Document binder**.
4. Enter a binder package name and description as shown.



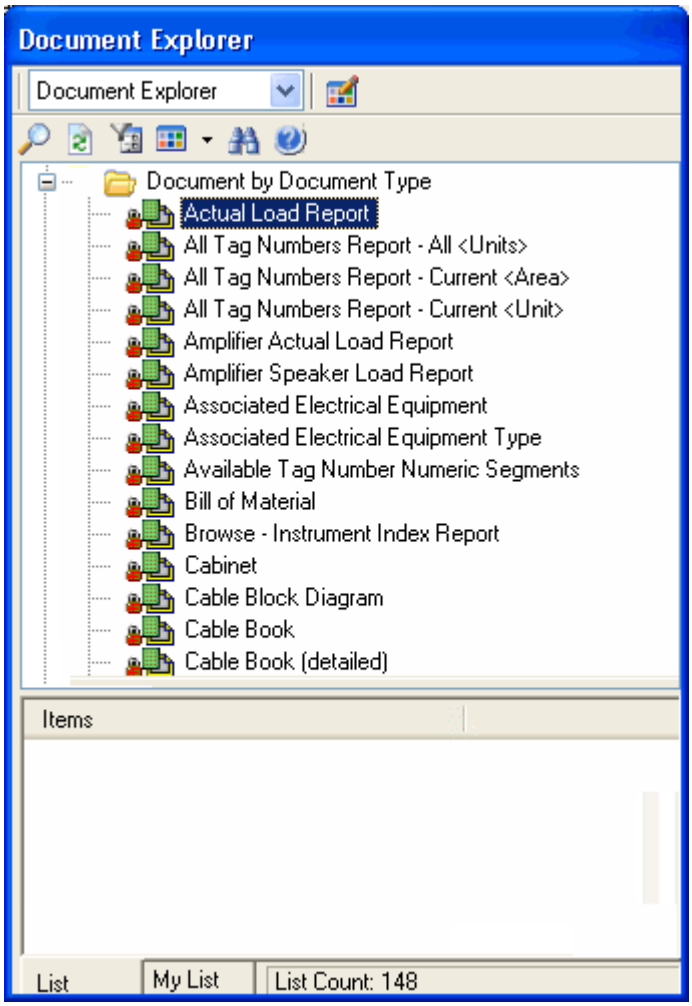
5. Click **OK**.

 The software automatically creates a document hierarchy for the General Document Binder package with the right pane displaying the column headers for the documents included in the binder package.

Add Calculation Reports to the Document Binder Package

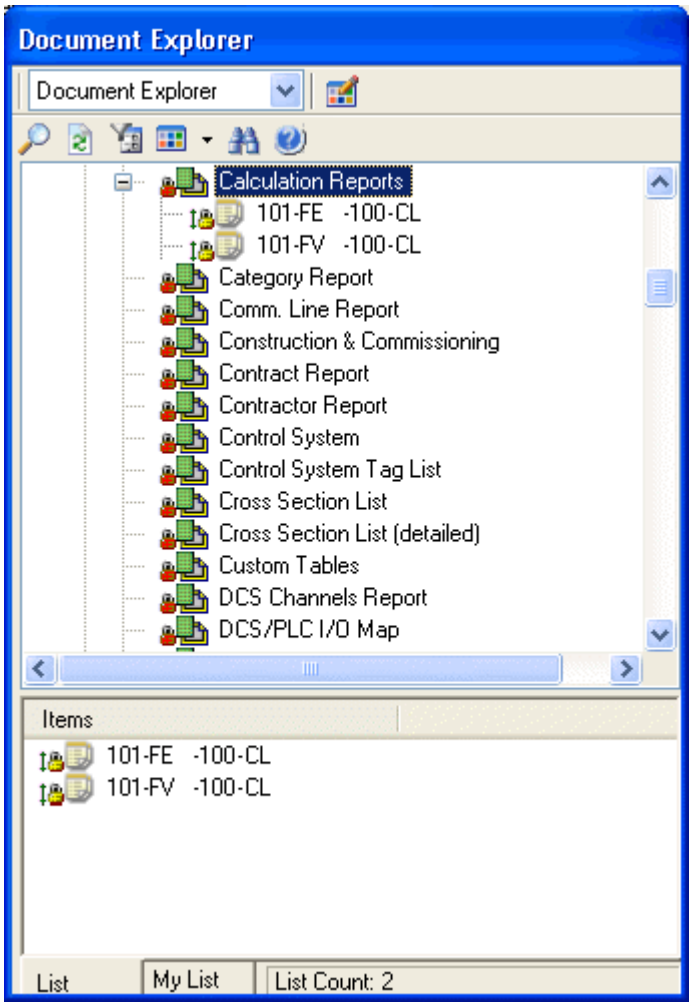
1. On the module toolbar, click to open the **Document Explorer** window.

2. Expand the **Document by Document Type** folder.



3. Scroll down and select **Calculation Reports**.

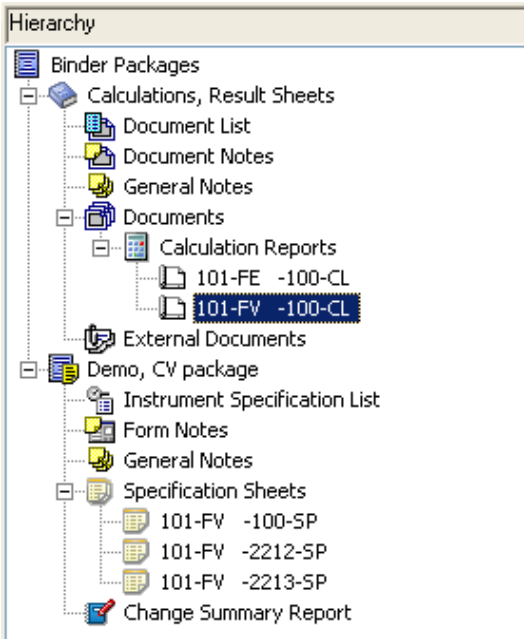
4. Expand the tree to display the available calculation reports as shown.



TIP Depending on the tags on which you performed calculations in the Calculation module, the list of sheets that appear on your screen may differ from those shown above.

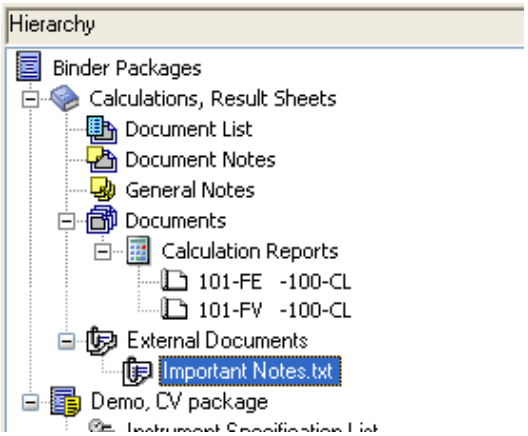
5. In the **Document Explorer** window, select document **101-FE -100-CL** and drag it to the **Documents** folder in the **Calculations, Result Sheets** General Document Binder package that you created.
6. Repeat the previous steps to add calculation sheet **101-FV -100-CL** to the **Calculations, Result Sheets** binder package.
7. Click again to close the **Document Explorer** window.

8. View the documents that were assigned to the General Document Binder package.



Assign External Documents to the Document Binder Package

1. In the **Hierarchy** tree view, expand the binder package folder **Calculations, Result Sheets**.
2. Right-click the **External Documents** folder, and on the shortcut menu, click **Associate External Documents**.
3. On the dialog box that opens, from the **Files of type** list, select **Document Files**.
4. Navigate to a file with extension .txt, .pdf, or .doc.
5. Select the desired file and click **Open**.
6. Expand the **External Documents** folder and right-click the document that you have just added, as shown in the example.

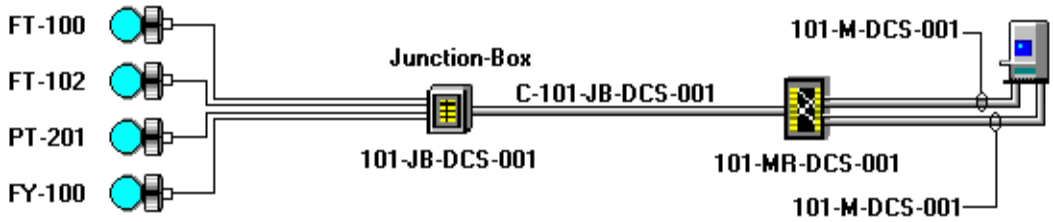


7. On the shortcut menu, click **Open** to open the document using the appropriate program.

SECTION 10

Performing Wiring Operations

Now you are going to learn how to create the necessary panels and cables and make the connections for the control loops. The diagram below is an example of a wiring scheme.



CAUTION Make sure that in the Administration module **Plant Properties** dialog box, you have cleared the **Do not propagate wire tag names** check box. For details, see *Create a Plant Hierarchy* (on page 25).

Understanding Concepts and Presentation of Panel and Cable Wiring

You can create all your panel and cable wiring from the **Domain Explorer**, which opens automatically after starting SmartPlant Instrumentation. If for some reason the **Domain Explorer** is not running, press **F7** to open it. Since it is possible to perform most wiring operations from the **Domain Explorer**, you do not need to start the Wiring module at this time.

Let us now examine how SmartPlant Instrumentation arranges the wiring items in the **Domain Explorer**.

Panels and their sub-items are arranged in the tree view of the **Domain Explorer** according to panel location and panel category. Therefore, you can find the same panel and its sub-items in both the **Panel by Location** and **Panel by Category** folders of the **Domain Explorer**.

Panels are classified according to their category. Click the **Panel by Category** folder to display the panel category folders. These folders will hold all your panels according to their categories. You cannot delete these folders, nor can you create new ones. Note that each panel category is represented by a particular icon to make it easily recognizable. The panel categories are as shown.

 — Junction box

 — Plug-and-socket junction box

 — Marshaling rack

 — Cabinet

 — Device panel

 — Plug-and-socket device panel

 — DCS (Distributed Control System) cabinet

 — PLC (Programmable Logical Controller) cabinet

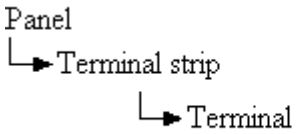
1. In the **Domain Explorer**, click **Panels by Category** > **Device Panels**.
2. Select a device panel.
3. Click the device panel to display the existing sub-items. Currently the next child item is a terminal strip.
4. Double-click the terminal strip to expand it and display its child items.

NOTE The **Telecom Panels** folder contains a separate group of panels that are specific to Telecom only. These panels are not dealt with in this tutorial.

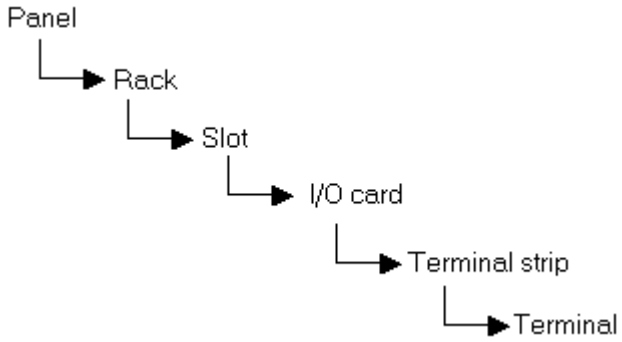
Panels by Category Hierarchy

Panels can contain a whole range of child items. These child items can be terminal strips, racks, wiring equipment, and so forth. There is a lot of flexibility in creating a panel hierarchy. There is no rigid structure like "panel – terminal strip – terminal" that limits your wiring design. You can create various wiring structures as you require. To create a new panel, right-click the **Panels by Category** folder, click **New** followed by a suitable command on the sub-menu. In our example, we have three types of panels (a junction box, a marshaling rack, and a DCS). We will make use of the following structures as shown.

For the junction box and marshalling rack:

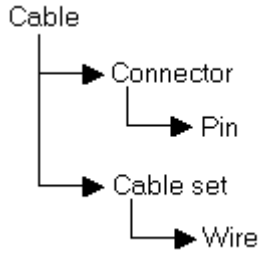


For the DCS panel:



Cable Hierarchy

Cables are arranged in the **Cables** folder. Cross wiring cables are arranged in the **Cross Cables** folder. To create a new cable, right-click the **Cables** folder, click **New** followed by a suitable command on the sub-menu. You can create various child items under cables. Here is one possible structure:



Task 1 — Creating Reference Panels

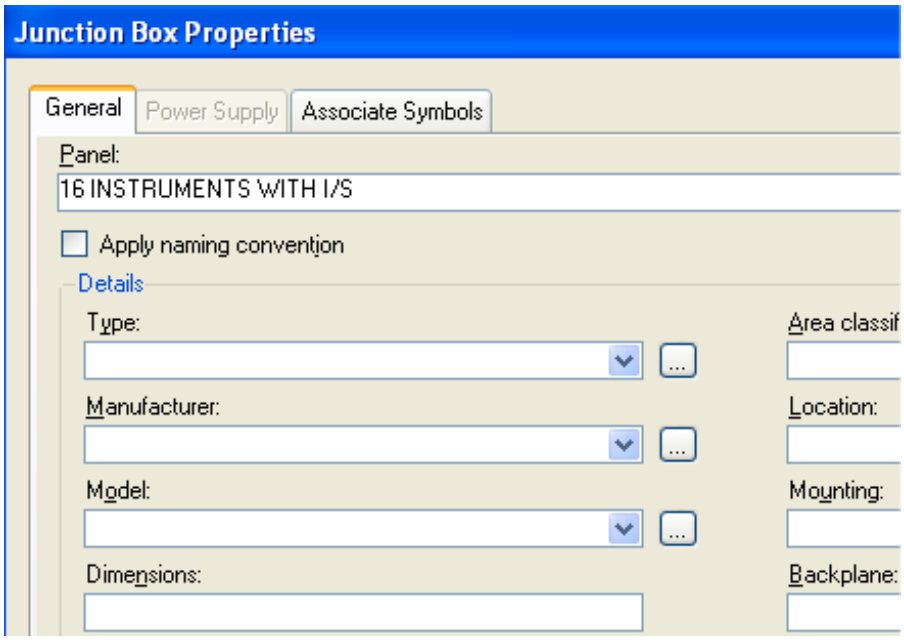
Reference items are typical wiring configurations that you can use for rapid creation of various wiring items. You use the **Reference Explorer** to create and manage all your reference items.

We strongly recommend that you create as many reference items in the **Reference Explorer** as possible. Try to create a reference item for each possible configuration that you may have in your domain. You can then drag this reference item from the **Reference Explorer** to the **Domain Explorer** and thus create the item that you require on the fly.

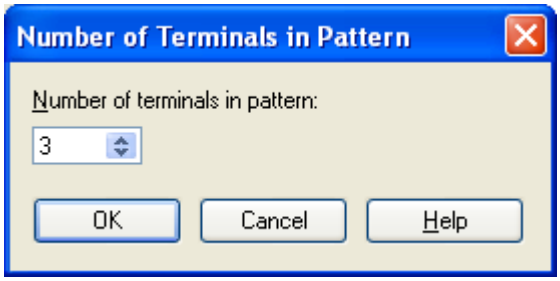
Create Reference Junction Boxes and Terminal Strips

1. Press **F8** to open the **Reference Explorer**.
2. Click the **Panels** folder.
3. Right-click the **Junction Boxes** folder, and on the shortcut menu, click **New** > **Junction Box (Generic)**.

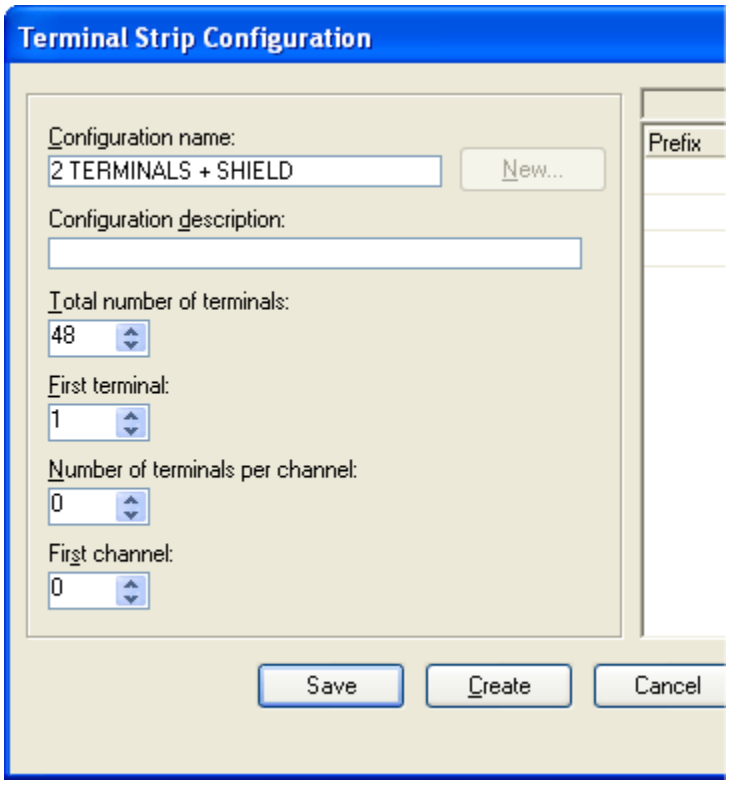
4. On the **Junction Box Properties** dialog box, type the panel name as follows.



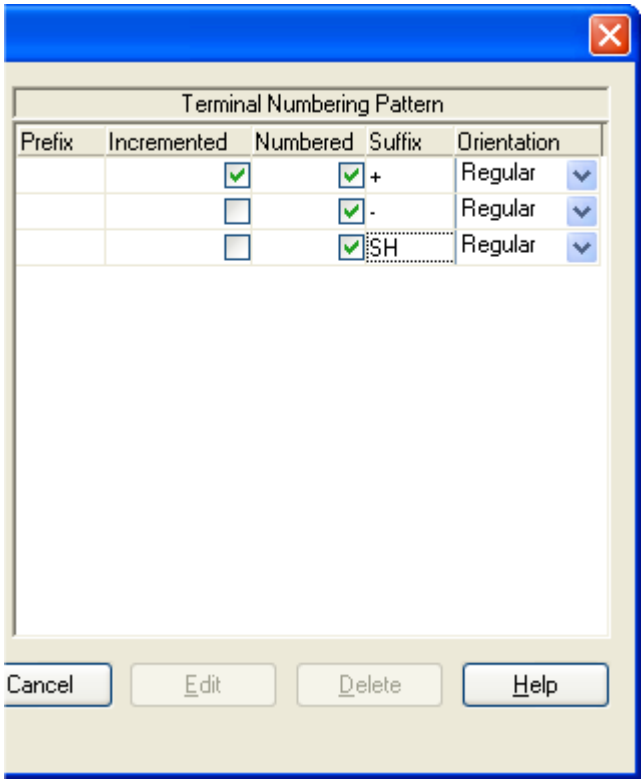
5. Click **OK**.
6. To view the new junction box in the **Reference Explorer**, double-click the **Junction Boxes** folder to expand it.
7. To add a terminal strip, right-click the new junction box and on the shortcut menu, c**lick New** > **Terminal Strip.**
8. On the **Terminal Strip Configuration** dialog box, click **New**.
9. On the **Number of Terminals in Pattern** dialog box, enter **3** and click **OK**.



10. Define the configuration of the new terminal strip as shown.



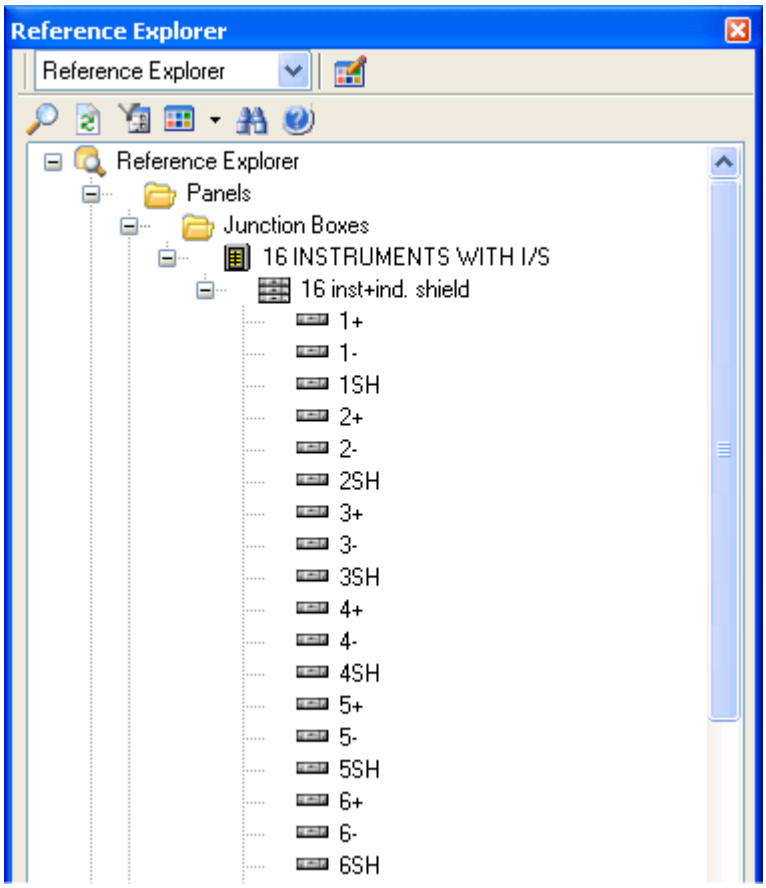
11. Define the terminal numbering as shown.



12. Click **Save** to save the configuration, then click **Create** to create the terminal strip.
13. On the **Terminal Strip Properties** dialog box, click in the **Terminal strip** field and name the terminal strip **16 inst+ind. shield** and then click **OK**.

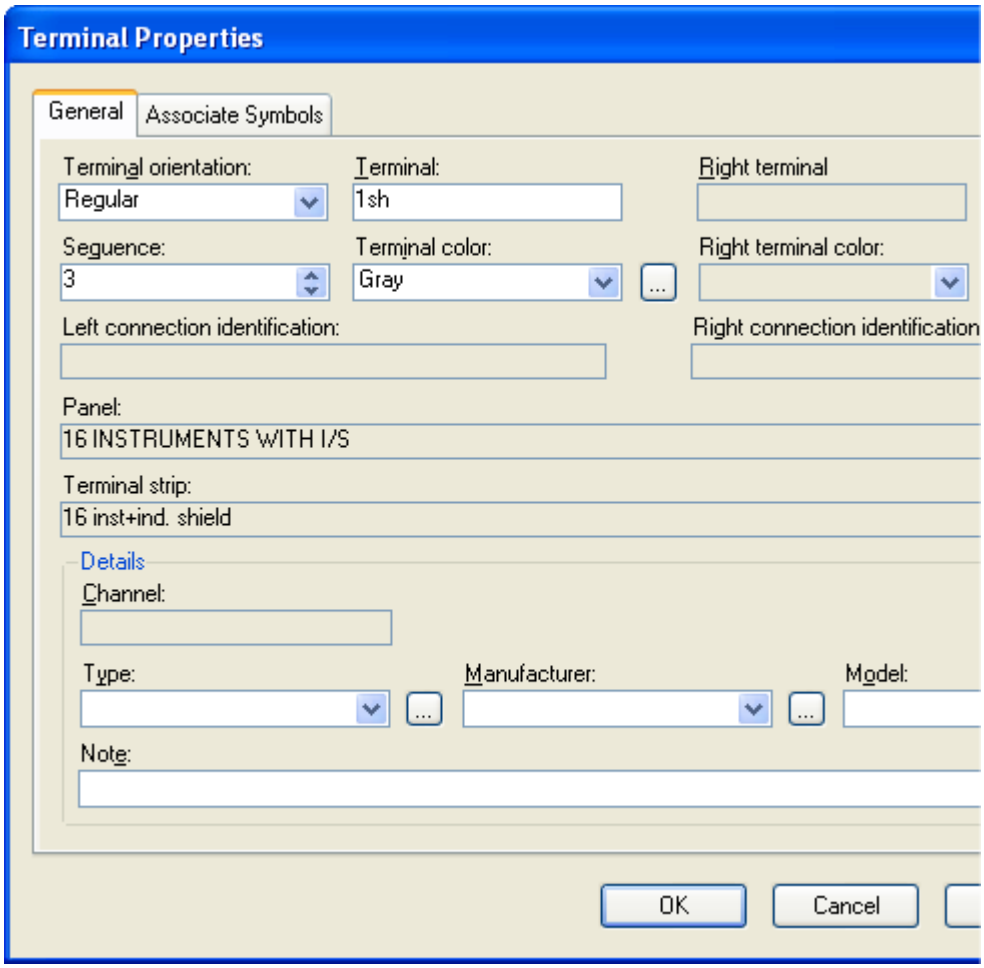
Edit Terminals

1. In the **Reference Explorer**, expand the panel **16 INSTRUMENTS WITH I/S**.



2. Scroll down to the third terminal of the terminal strip (the one labeled **1SH**) and select it.
3. Right-click this terminal and then on the shortcut menu, click **Properties**.

4. On the **Terminal Properties** dialog box, under **Terminal**, change **1SH** to **1sh**.



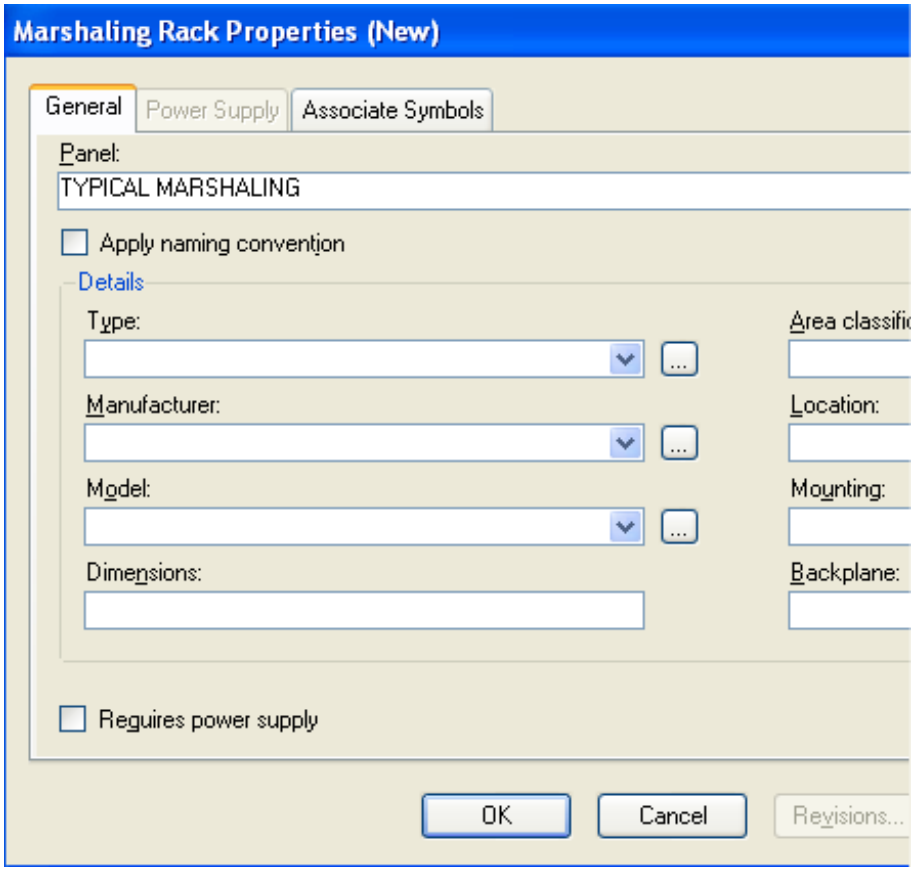
5. Click **OK** and observe the change in the **Reference Explorer**.
6. Now change the value back to **1SH** again.

NOTE Terminal colors are not shown in the **Domain Explorer** and **Reference Explorer**. You will see them later when you open the **Connection** window.

Create a Reference Marshaling Rack

1. In the **Reference Explorer**, expand the **Panels** folder.
2. Right-click the **Marshaling Racks** folder, and on the shortcut menu, click **New** > **Marshaling Rack**.

3. On the **Marshaling Rack Properties** dialog box, in the **Panel** field, type **TYPICAL MARSHALING** (you can leave the remaining fields blank).



4. Click **OK** to create this marshaling rack.

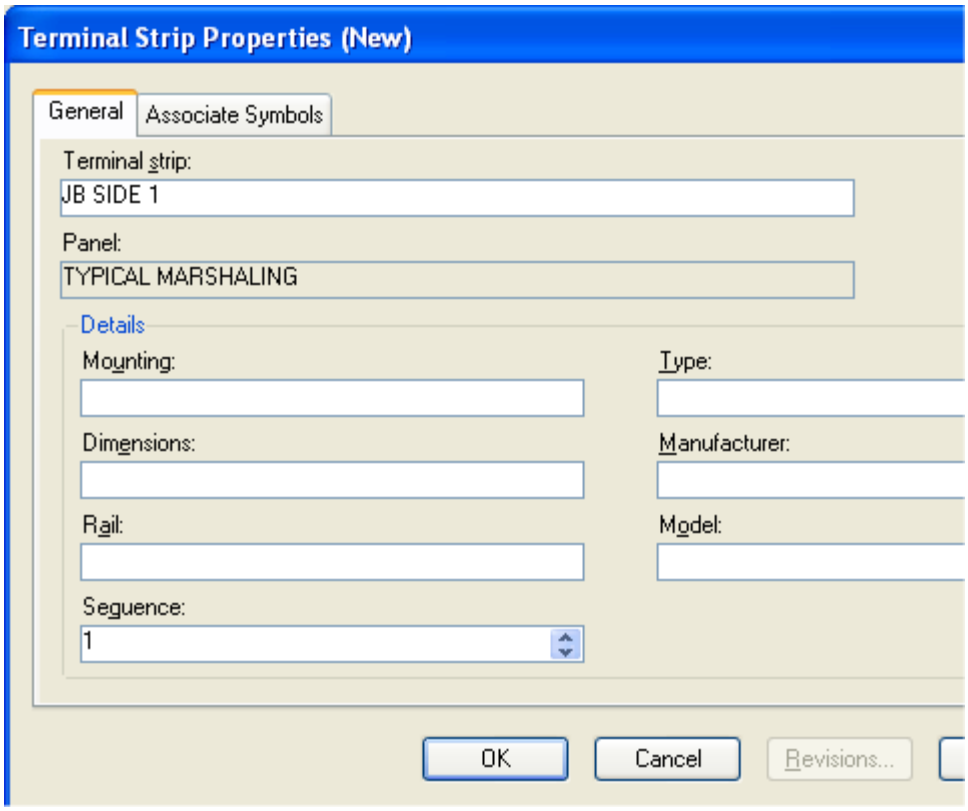
Task 2 — Creating Terminal Strips within a Marshaling Rack

You will now create the following terminal strips within this marshaling rack:

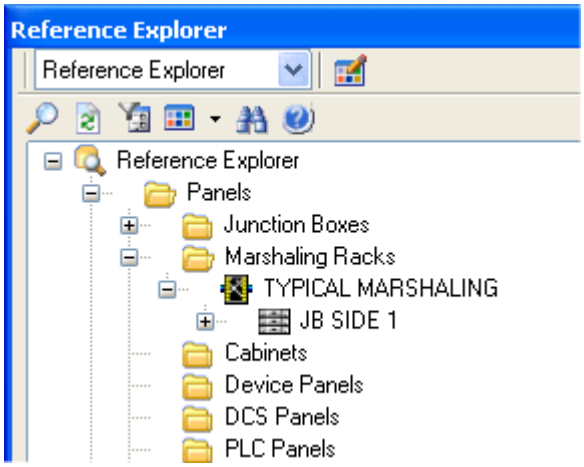
- JB SIDE-1 and JB SIDE-2 to accommodate your junction box cables (field side).
- DCS SIDE-AI to accommodate your DCS side cable containing analog input signals (DCS side).
- DCS SIDE-AO to accommodate your DCS side cable containing analog output signals.

Create JB SIDE-1

1. In the **Reference Explorer**, expand the **Panels** folder.
2. Right-click marshaling rack **TYPICAL MARSHALING**, and on the shortcut menu click **New** > **Terminal Strip**.
3. On the **Terminal Configuration** dialog box, select the **2 TERMINALS + SHIELD** configuration (this is the same configuration you used for the junction box).
4. Click **Create** to open the **Terminal Strip Properties** dialog box.
5. Complete the **Terminal Strip Properties** dialog box as shown and click **OK**.



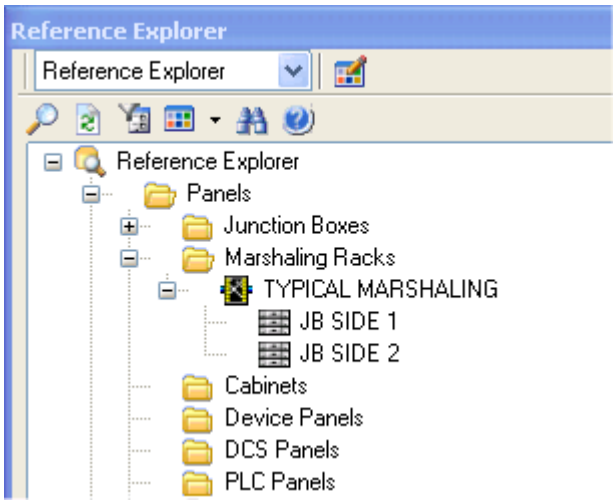
6. In the **Reference Explorer**, expand the marshaling rack **TYPICAL MARSHALING** to view the newly created terminal strip.



Create Strip JB SIDE-2 by Duplicating JB SIDE-1

1. In the **Reference Explorer,** right-click **JB SIDE-1**.
2. On the shortcut menu, click **Duplicate**.
3. On the **Terminal Strip Properties** dialog box, change the terminal strip name to **JB SIDE-2**.

 TIP Observe that the value of the sequence is **2**.
4. Click **OK** to accept the new values and close the dialog box.



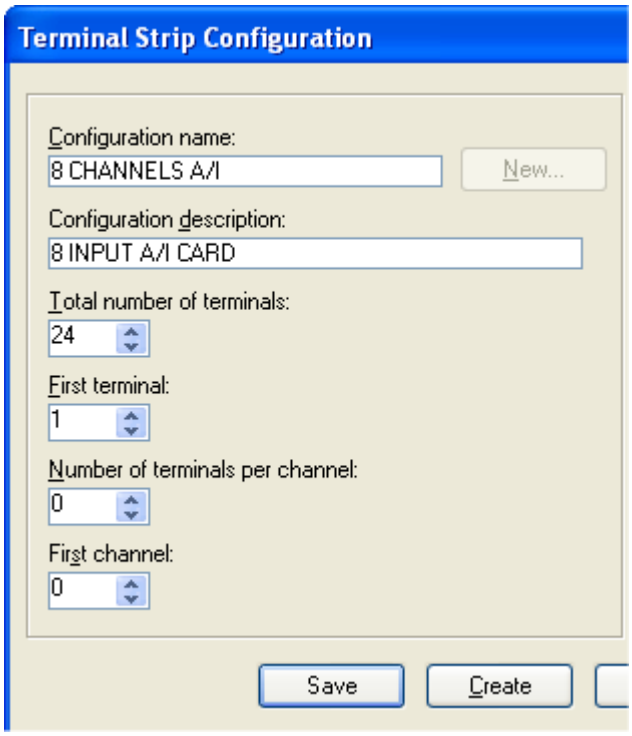
Create DCS Side Terminal Strip DCS SIDE - AI

You will now create DCS side terminal strips that correspond to analog input cards and analog output cards.

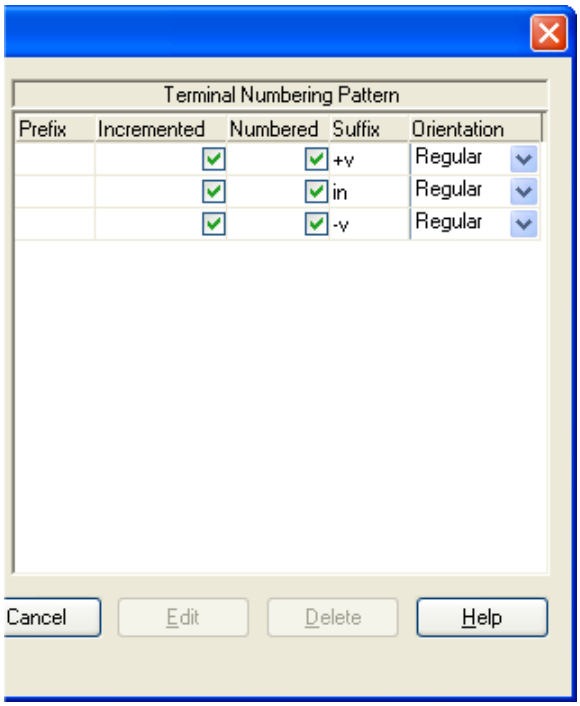
1. In the **Reference Explorer**, right-click marshaling rack **TYPICAL MARSHALING**, and on the shortcut menu click **New** > **Terminal Strip**.

 The **Terminal Strip Configuration** dialog box opens. You will now create a new terminal configuration and a new terminal strip.

2. Click **New** and on the **Number of Terminals in Pattern** dialog box, enter **3**.
3. Click **OK**.
4. On the **Terminal Strip Configuration** dialog box, define the terminal configuration as shown.

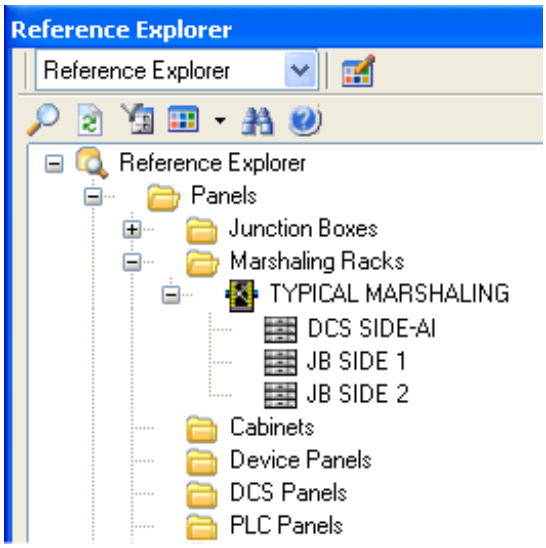


5. Define the terminal numbering pattern as shown.



6. Click **Save**.
7. Click **Create** to open the **Terminal Strip Properties** dialog box where you create the new terminal strip.
8. On the **Terminal Strip Properties** dialog box, under **Terminal strip**, type **DCS SIDE-AI** and click **OK**.

 Terminal strip **DCS SIDE-AI** now appears in the **Reference Explorer** under marshaling rack **TYPICAL MARSHALING**.



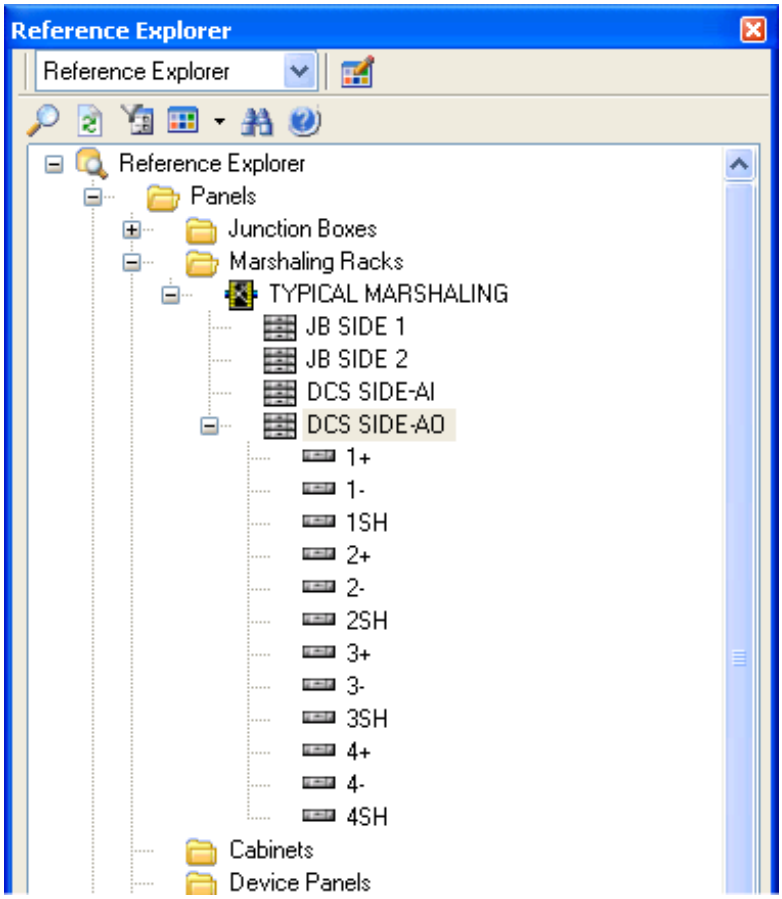
9. Click terminal strip **DCS SIDE-AI** to view the terminals on that strip.

Create DCS Side Terminal Strip DCS SIDE-AO by Duplicating JB SIDE-1

1. In the **Reference Explorer**, expand the hierarchy: **Panels** > **Marshaling Racks** > **TYPICAL MARSHALING**.
2. Right-click terminal strip **JB SIDE-1** and then on the shortcut menu, click **Duplicate**.
3. On the **Terminal Strip Properties** dialog box, change the terminal strip name to **DCS SIDE-AO**.
4. Click **OK**.

 Terminal strip **DCS SIDE-AO** now appears in the **Reference Explorer** under marshaling rack **TYPICAL MARSHALING**.
5. Expand **DCS SIDE-AO** as we want to have a 4-channel AO terminal strip.
 You will now delete the surplus terminals in terminal strip **DCS SIDE-AO**:
 a. In the **Items** pane of the **Reference Explorer**, select terminal **5+** and while holding down **Shift**, select Terminal **16SH**; this action selects all the terminals between **5+** and **16SH**.
 b. On your keyboard, press **Delete**.
 c. When prompted, click **Yes** to confirm the deletion.

 Only 12 terminals (from **1+** to **4SH**) now remain in terminal strip **DCS SIDE-AO**:



NOTE The same result can be achieved by creating a new terminal strip configuration using the **New** > **Terminal Strip** command on the shortcut menu.

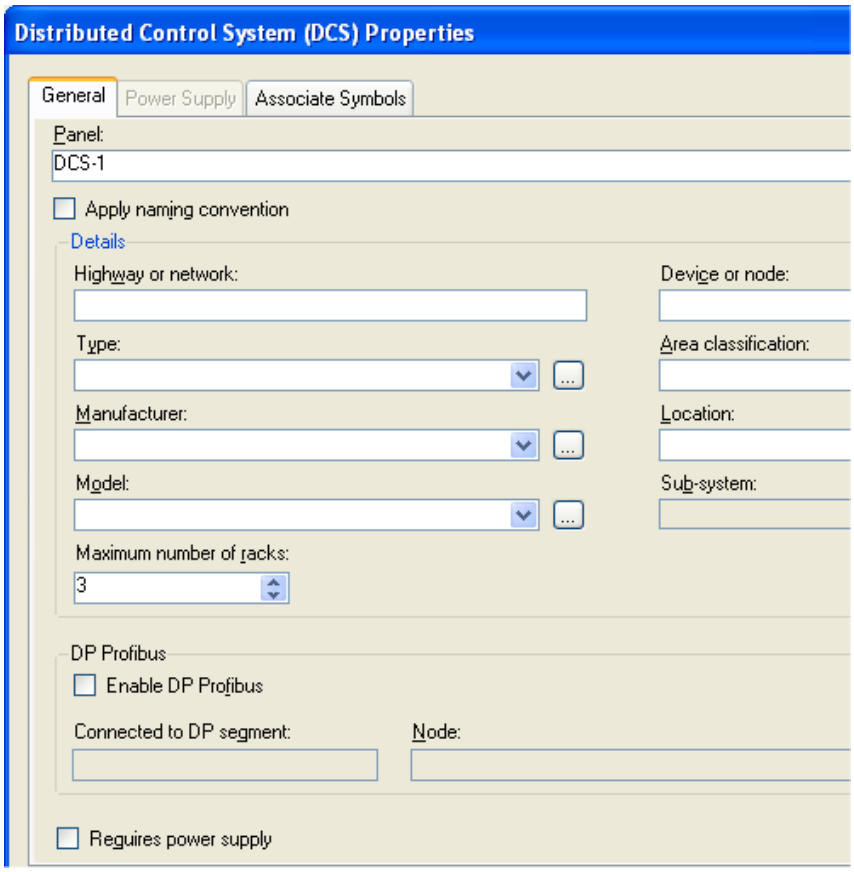
You have now completed the creation of marshaling rack **TYPICAL MARSHALING** with its internal terminal strip.

Task 3 — Creating a Reference DCS Panel

You will now create a DCS panel that contains 3 racks. The first one will contain one analog input card and one analog output card.

Create a Reference DCS Panel

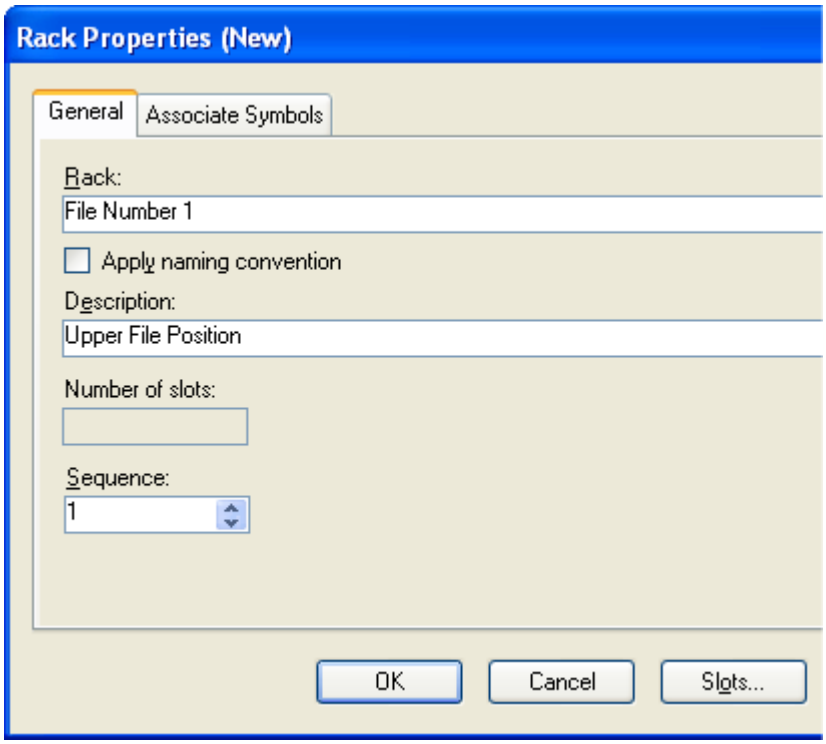
1. In the **Reference Explorer**, expand the **Panels** folder.
2. Right-click the **DCS Panels** folder and then on the shortcut menu, click **New** > **DCS Panel**.
3. On the **Distributed Control System (DCS) Properties** dialog box, under **Panel** type **DCS-1**.
4. Under **Maximum number of racks**, enter **3**.



5. Click **OK**.

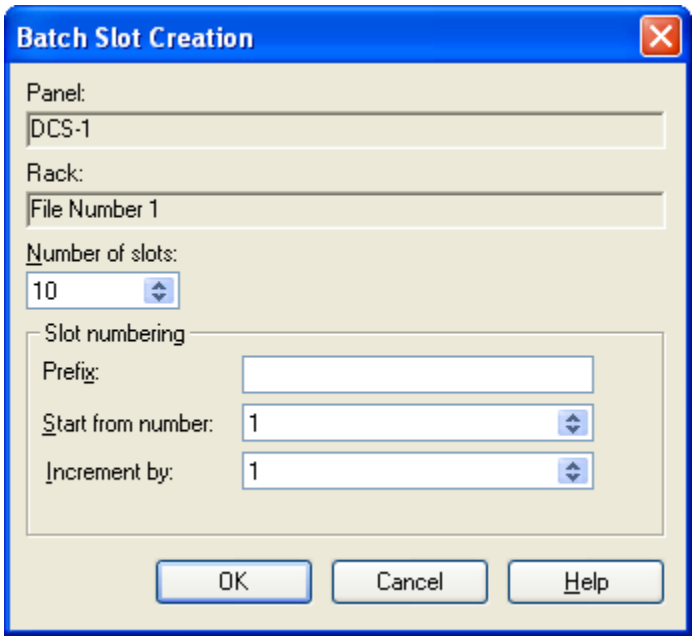
Create a Rack in the New Reference DCS Panel

1. In the **Reference Explorer**, right-click the new **DCS-1** panel.
2. On the shortcut menu, click **New** > **Rack**.
3. On the **Rack Properties** dialog box, enter the information as shown.

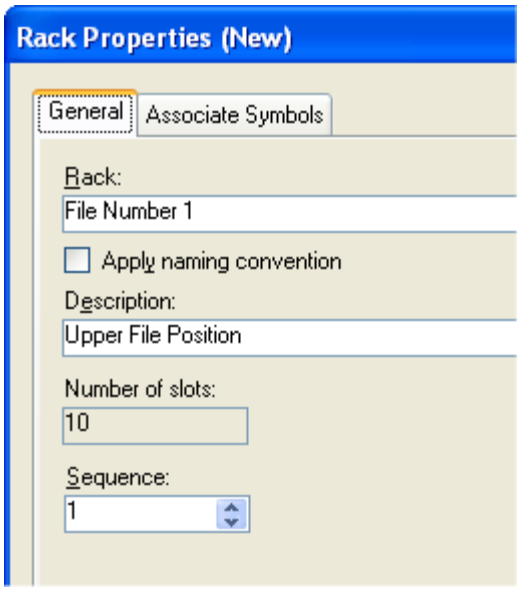


4. Click **Slots**.
5. In the prompt message click **OK** to save the data on the **Rack Properties** dialog box.
6. On the **Batch Slot Creation** dialog box, under **Number of slots**, type **10**.
7. Under **Start from number**, type **1**.

8. Under **Incremented by**, type **1**.



9. Click **OK** to return to the **Rack Properties** dialog box. Note that there are now 10 slots defined in the new rack.

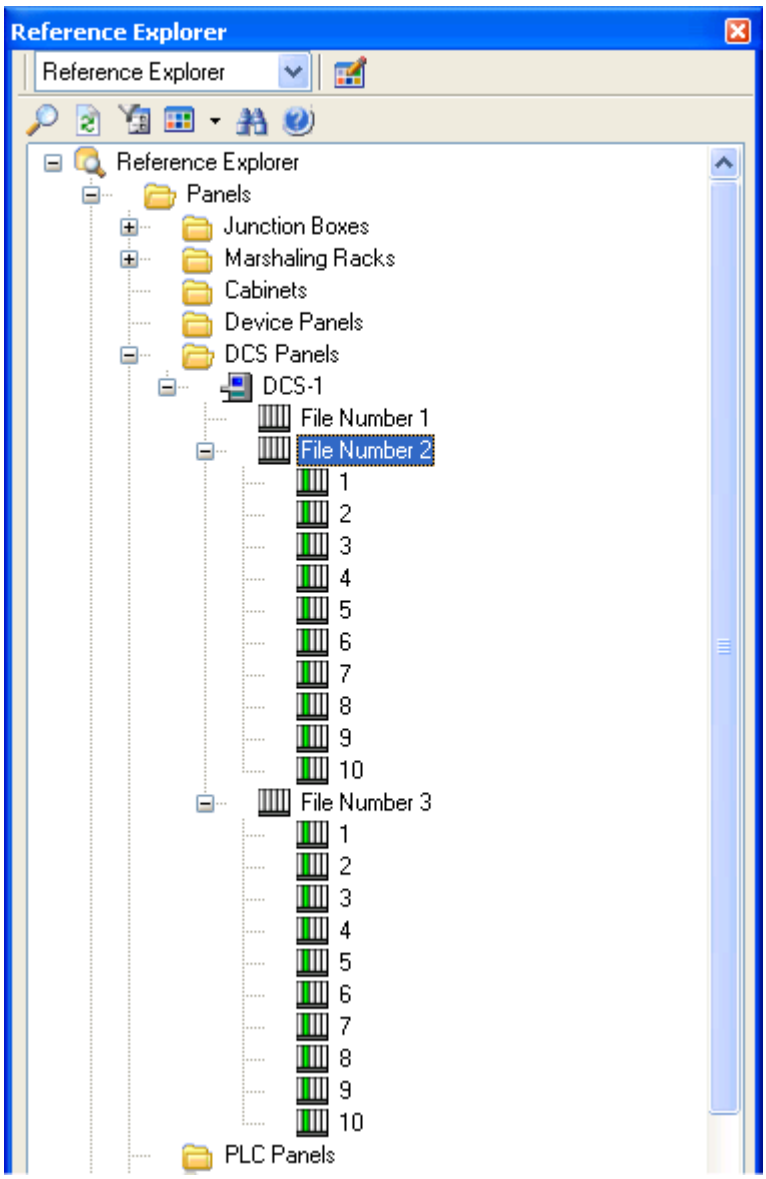


10. Click **OK** to close the **Rack Properties** dialog box.

Create Two Additional Racks in the New Reference DCS Panel

1. Create two more racks in the **DCS-1** panel by duplicating the new rack that you just created.
2. Name the new racks **File Number 2** and **File Number 3**.
3. Make sure that the new racks have ten slots each.

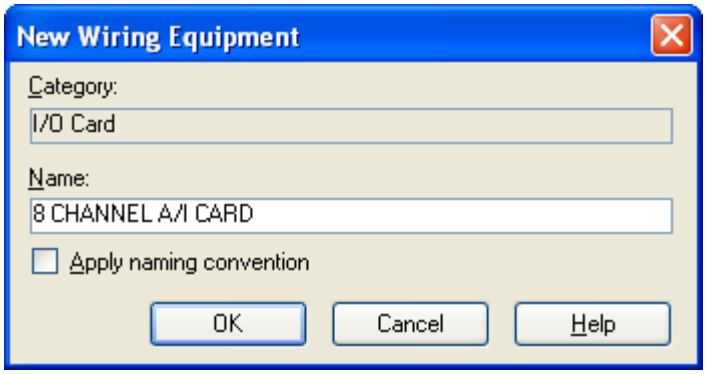
 After creating the new racks, the hierarchy of DCS-1 should appear as shown:



Create an Analog Input I/O Card within the New Reference DCS Panel

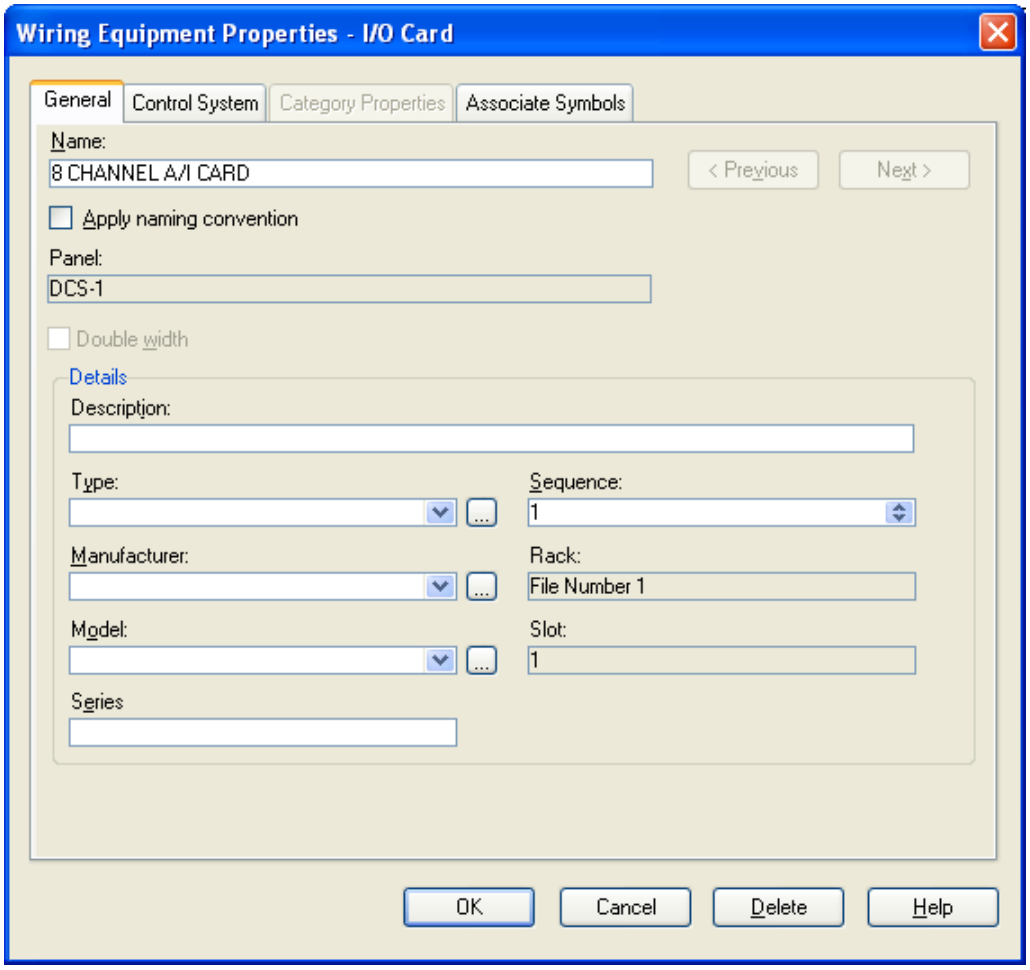
After creating the required racks and slots in the new reference DCS panel, you can now create an analog input I/O card.

1. In the **Reference Explorer**, click the **DCS-1** panel to display all its sub-items.
2. Right-click slot **1** under **File Number 1** and then on the shortcut menu, click **New** > **I/O Card**.
3. On the **New Wiring Equipment** dialog box, type the new I/O card name **8 CHANNEL A/I CARD**.

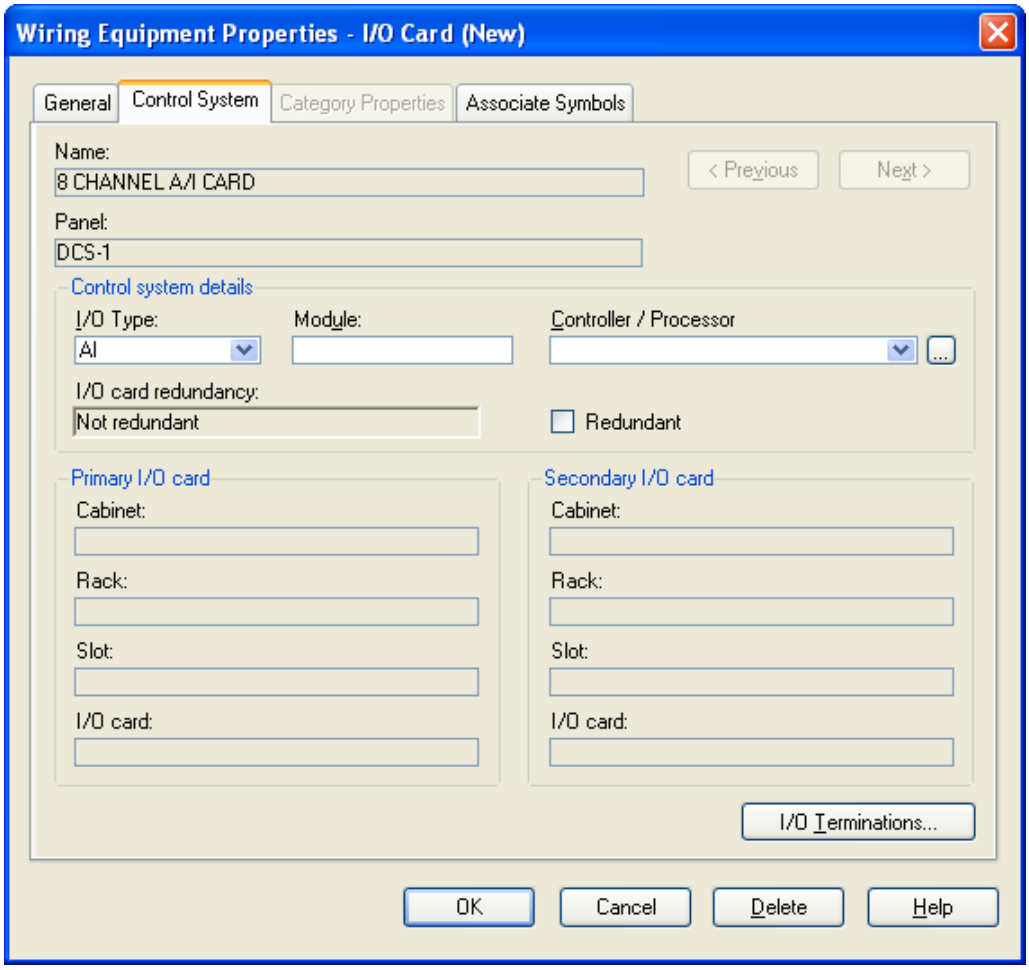


4. Click **OK**.

5. On the **Wiring Equipment Properties – I/O Card** dialog box, enter values for the I/O card properties as shown.



6. Click the **Control System** tab and under **Control system details**, from the **I/O type** list, select **AI**.

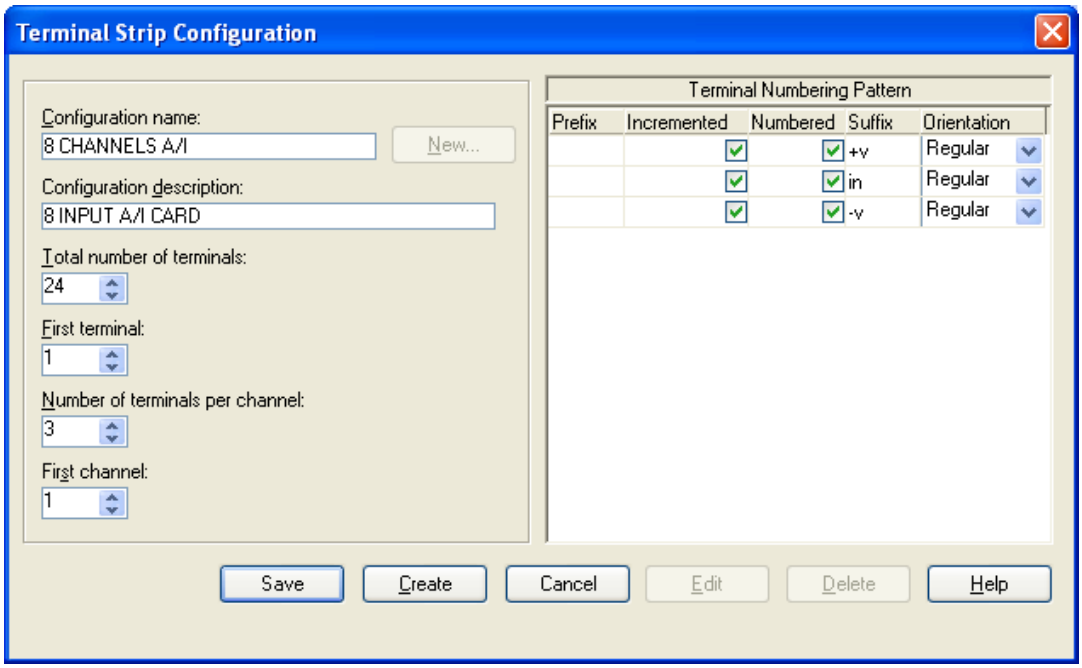


7. Click **OK** to accept all the values and close the **Wiring Equipment Properties – I/O Card** dialog box.

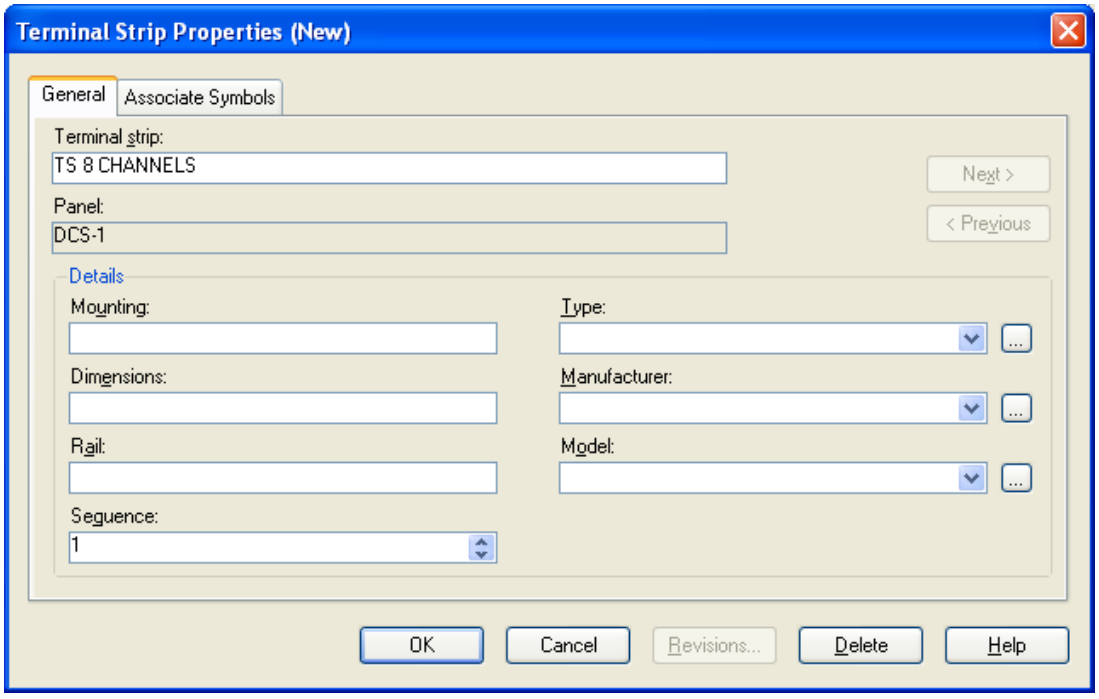
Create a Terminal Strip with I/O Channels

1. In the **Reference Explorer**, expand the following hierarchy:
 DCS-1 > **File number 1** > **1** (slot).
2. Right-click I/O card **8 CHANNEL A/I CARD**, and then on the shortcut menu click **New** > **Terminal Strip**.
3. On the **Strip Terminal Configuration** dialog box, from the **Configuration name** list, select **8 CHANNELS A/I**.
4. Click **Edit**.

5. Change the configuration as shown.



6. Click **Save**.
7. Click **Create** and then define the new terminal strip as shown.

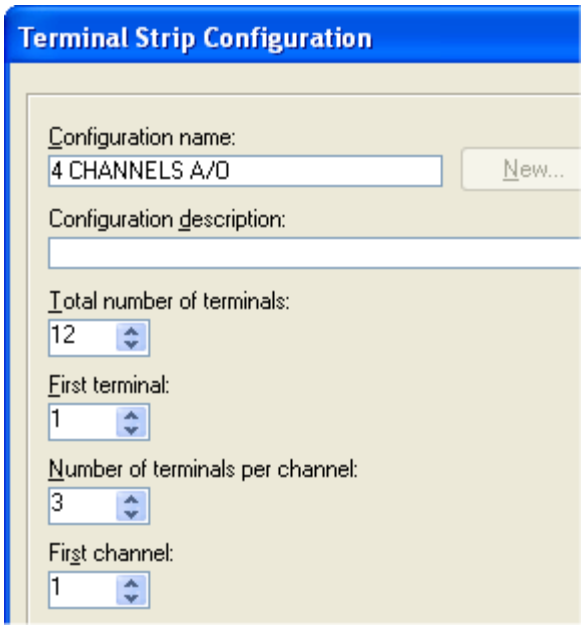


8. Click **OK**.

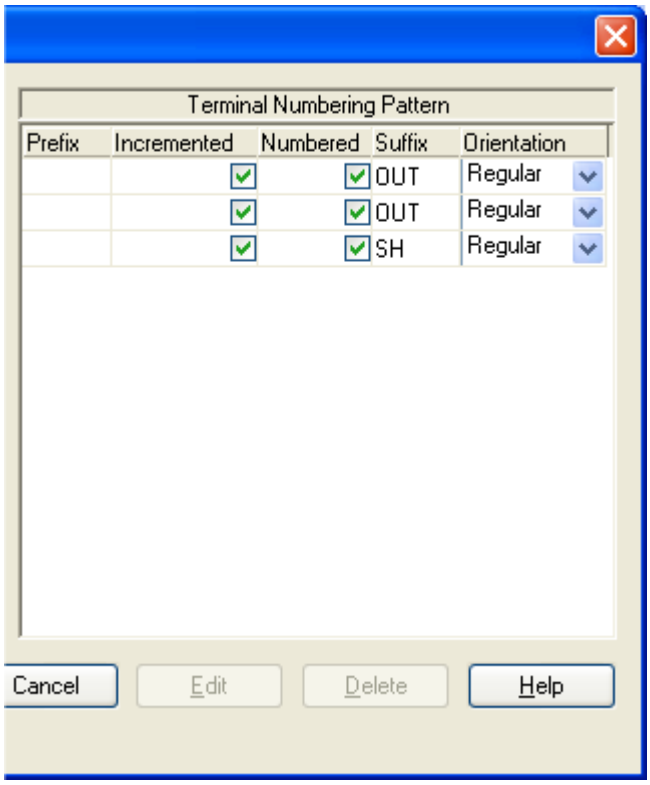
Create an Analog Output I/O Card and a 4-Channel Terminal Strip within the New Reference DCS Panel

After creating the required racks and slots in the new reference DCS panel, you can now create an analog input I/O card.

1. In the **Reference Explorer**, expand the panel **DCS-1** to display all its child items.
2. Under **File Number 1**, right-click **Slot 2** and then on the shortcut menu, click **New** > **I/O Card**.
3. On the **New Wiring Equipment** dialog box, type the new I/O card name **4 CHANNEL A/O CARD** and click **OK**.
4. On the **Wiring Equipment Properties – I/O Card** dialog box, on the **General** tab, accept the I/O card name.
5. Click the **Control System** tab and from the **I/O type** list, select **AO**.
6. Click **OK** to complete the creation of the new I/O card.
7. Right-click the new I/O card **4 CHANNEL A/O CARD** and on the shortcut menu, click **New** > **Terminal Strip**.
8. On the Terminal Strip Configuration dialog box, next to **Configuration name** click New.
9. On the **Number of Terminals in Pattern** dialog box, enter **3** and click **OK**.
10. Define the terminal configuration as shown.

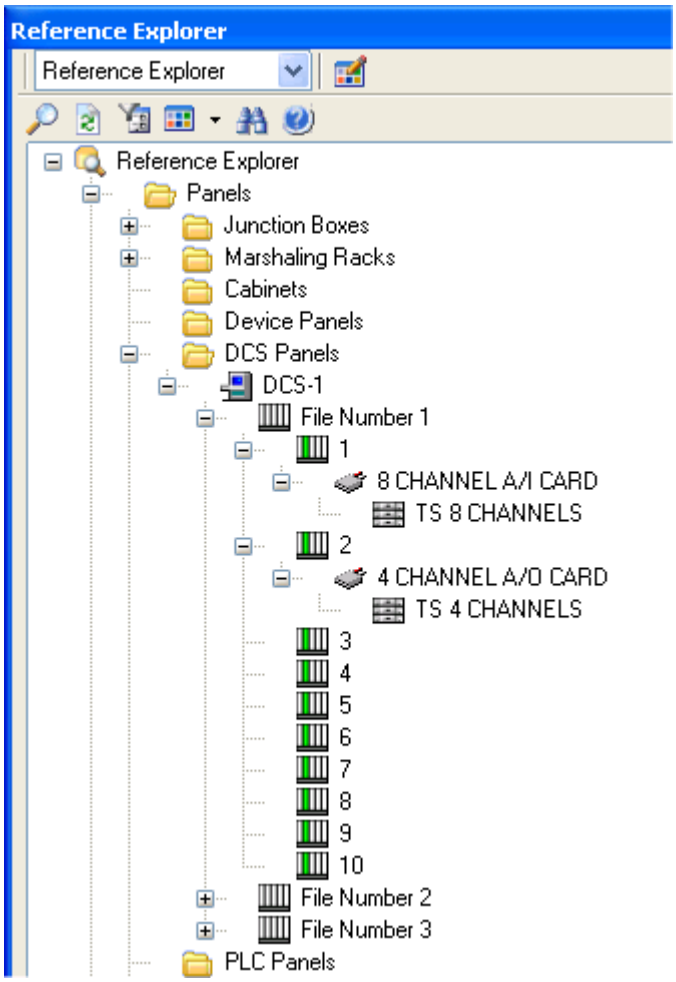


11. Define the terminal numbering pattern as shown.



12. Click **Save** and then **Create**.
13. On the **Terminal Strip Properties** dialog box, under **Terminal strip**, type **TS 4 CHANNELS** and click **OK**.

The **Reference Explorer** should now display the following items in the **DCS Panels** folder:



Task 4 — Copying the Reference Panels to the Domain Explorer

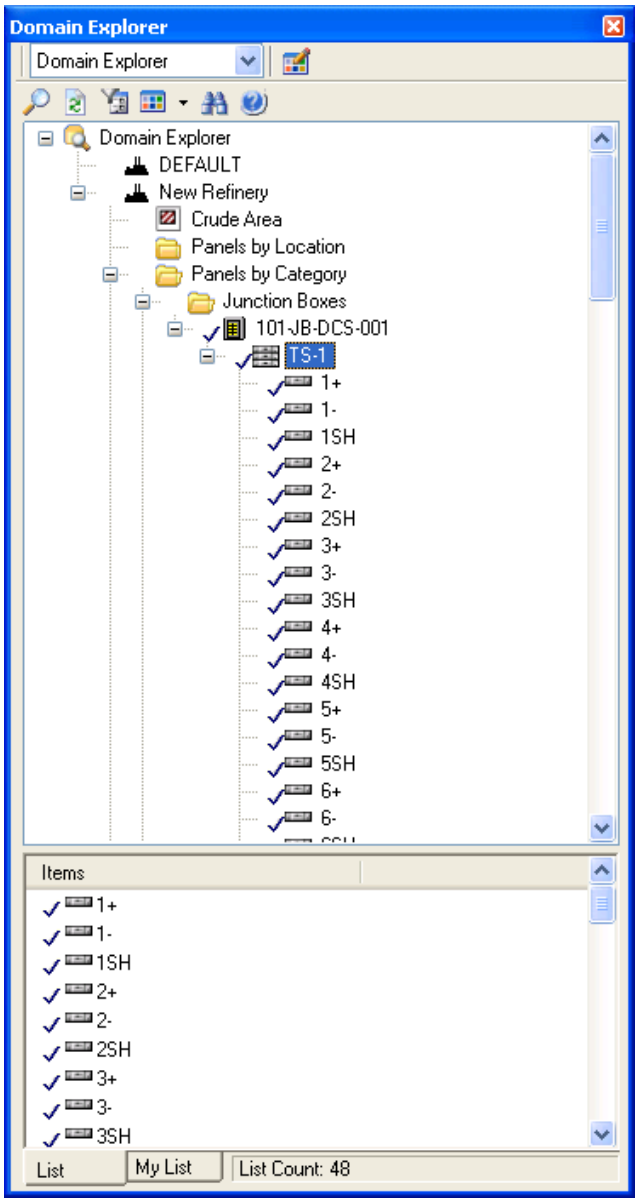
You will now make use of the reference panels in your plant.

Create a Plant Junction Box

1. Make sure that both the **Domain Explorer** and the **Reference Explorer** are open side by side. (Press **F7** to open the **Domain Explorer** and **F8** to open the **Reference Explorer**.)
2. In the **Domain Explorer**, expand the **Panels by Category** folder to display the **Junction Boxes** folder.
3. In the **Reference Explorer**, expand the **Panels** folder.
4. In the **Reference Explorer**, expand the **Junction Boxes** folder and then click junction box **16 INSTRUMENTS WITH I/S**.
5. Drag this junction box to the **Junction Boxes** folder under the **Panels by Category** folder in the **Domain Explorer**.

 NOTE When you release the mouse button to drop the junction box in the **Domain Explorer**, the software opens the **Junction Box Properties** dialog box.
6. On the **Junction Box Properties** dialog box, under the **Panel** field, type **101-JB-DCS-001** and click **OK** to close the dialog box.
7. In the **Domain Explorer**, expand junction box **101-JB-DCS-001**. Note that it contains the same sub-items as in the junction box that you created in the **Reference Explorer**.
8. Right-click terminal strip **16 inst+ind. shield**, and select **Properties** on the shortcut menu to open the **Terminal Strip Properties** dialog box.
9. In the **Terminal strip** field, type **TS-1** over the existing terminal strip name and click **OK**.

The renamed terminal strip appears in the **Domain Explorer** as shown:



Create a Plant Marshaling Rack

1. Based on the previous procedure, copy the reference marshaling rack **TYPICAL MARSHALING** to the **Marshaling Racks** folder under the **Panels by Category** folder in the **Domain Explorer**.
2. On the **Marshaling Rack Properties** dialog box, rename your plant marshaling rack to **101-MR-DCS-001**.

Rename the Terminal Strips in the Domain Explorer

1. Expand your new marshaling rack **101-MR-DCS-001** in the **Domain Explorer** to display its terminal strips.
2. Right-click terminal strip **DCS SIDE-AI** and then click **Properties** on the shortcut menu.
3. On the **Terminal Strip Properties** dialog box, under **Terminal strip**, type **TP-1** over the existing name.
4. Click **OK** to save the new definitions and close the screen.
5. Repeat steps 2 through 4 to rename the rest of the terminal strips as follows:

Old Reference Terminal Strip Name	New Plant Terminal Strip Name
DCS SIDE-AO	TP-2
JB SIDE-1	FT-1
JB SIDE-2	FT-2

Copy Your DCS from the Reference Explorer to the Domain Explorer

1. Drag **DCS-1** from the **Reference Explorer** to the **DCS Panels** folder in the **Domain Explorer**.
2. Rename **DCS-1** to **101-DCS-001**.
3. Expand **101-DCS –001**, slot **1** to display its I/O card and terminal strip.
4. Rename the I/O card to **8 CH AI** and the terminal strip to **TB 1**.
5. Expand slot **2** to display its I/O card and terminal strip.
6. Rename the I/O card to **4 CH AO** and the terminal strip to **TB1**.

Task 5 — Creating Reference Cables

You will now create your reference cables that will be later used in your plant. Reference cables are typical cable configurations based on manufacturer cable specifications.

You will create the following types of cable:

- A single-pair cable
- 16-pair with individual shields
- 8-pair with individual shields
- 4-pair with individual shields

TIP It is recommended that you create typical configurations of cables, sets and wires in the **Reference Explorer**. Doing so enables you to create numerous plant cables on the fly.

Create a Reference Single-Pair Cable

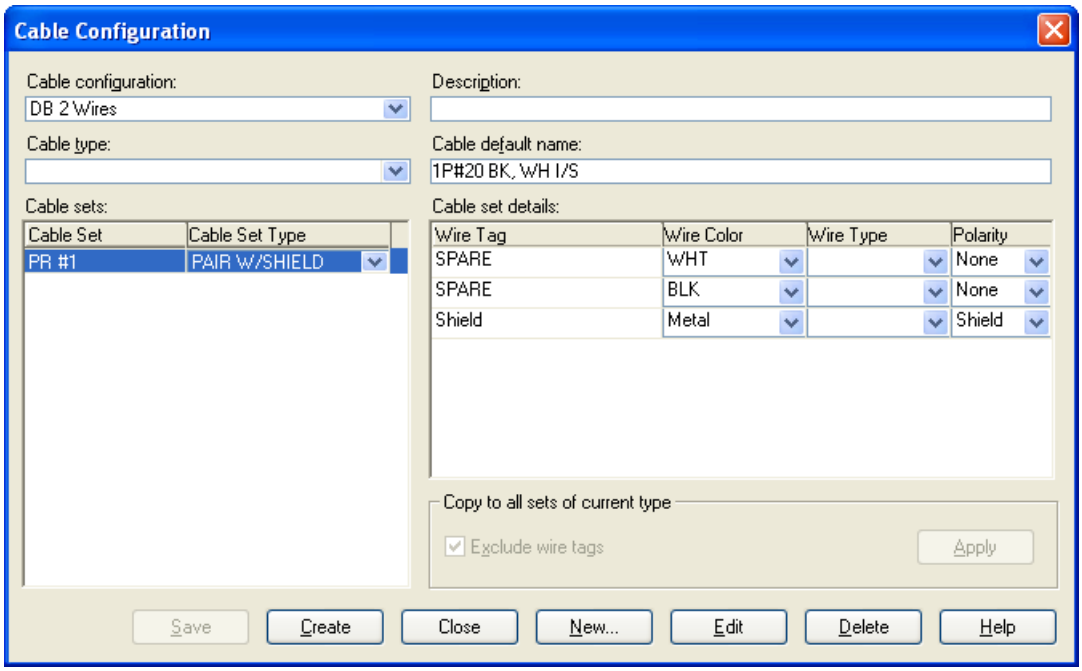
NOTE You have already used a single-pair cable in your instrument type profile to enable the system to automatically create field device objects connected to a single-pair cable. Nevertheless, for tutorial purposes, you will learn how to create reference cables.

To create a new cable, you need to do the following:

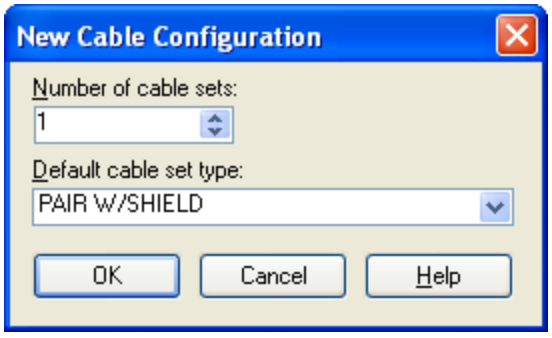
1. Add a cable.
2. Select a cable configuration.
3. Use a configuration to create a cable with its associated cable sets and wires.

Creating a new Cable

1. Press **F8** to open the **Reference Explorer**.
2. Right-click the **Cables** folder and then on the shortcut menu, click **New** > **Cable**.

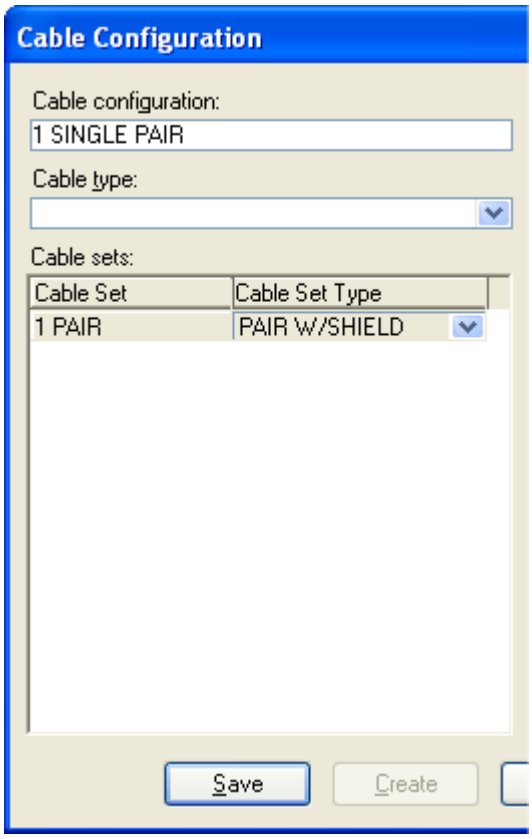


3. On the **Cable Configuration** dialog box, click **New** to create a new cable configuration.
4. On the **New Cable Configuration** dialog box that opens, enter data as shown:

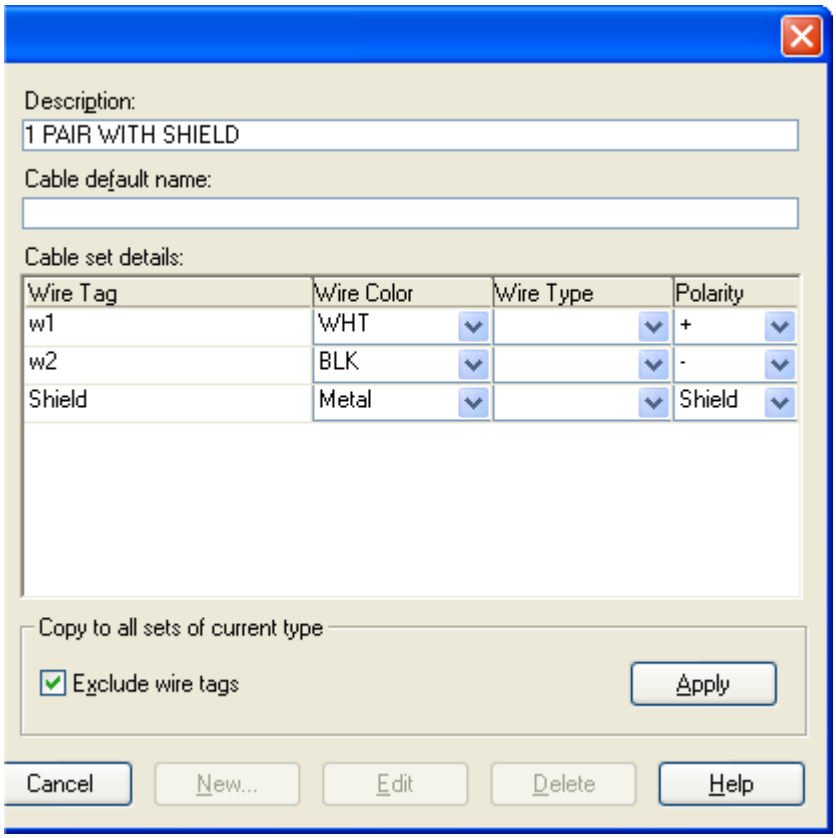


5. Click **OK** to return to the **Cable Configuration** dialog box.

6. Complete the data as shown below:

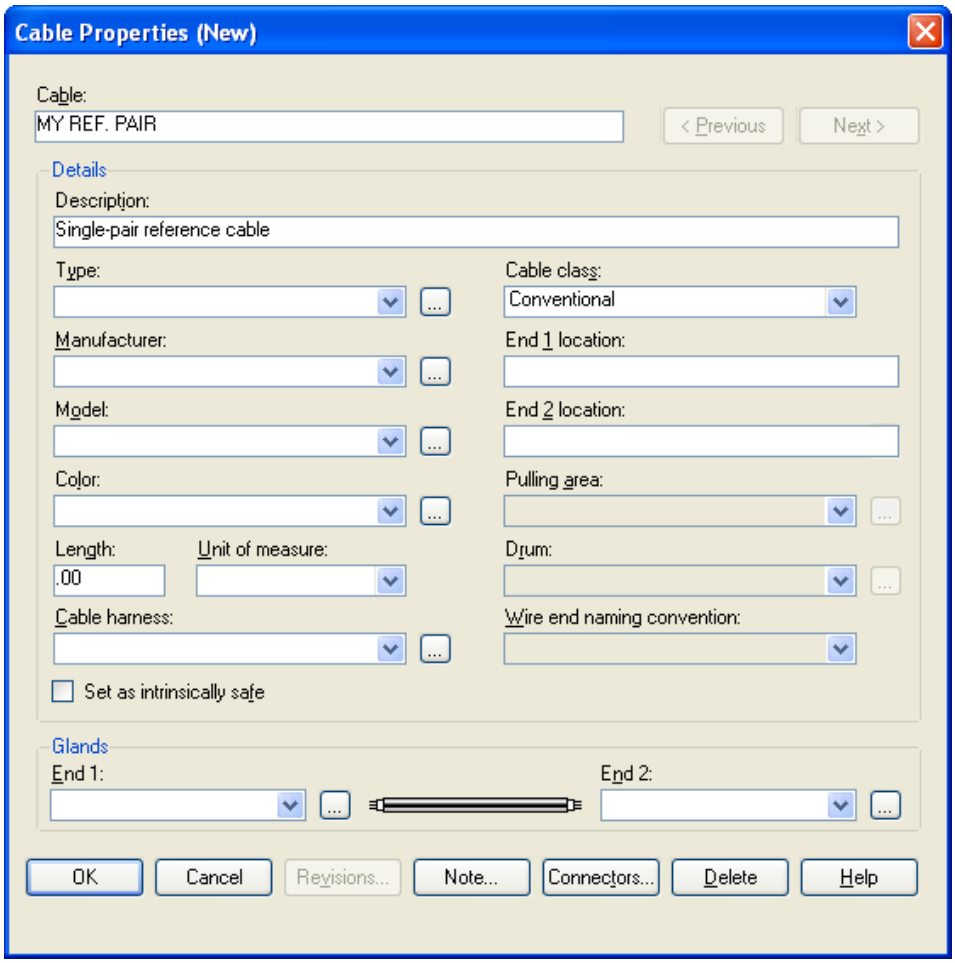


and



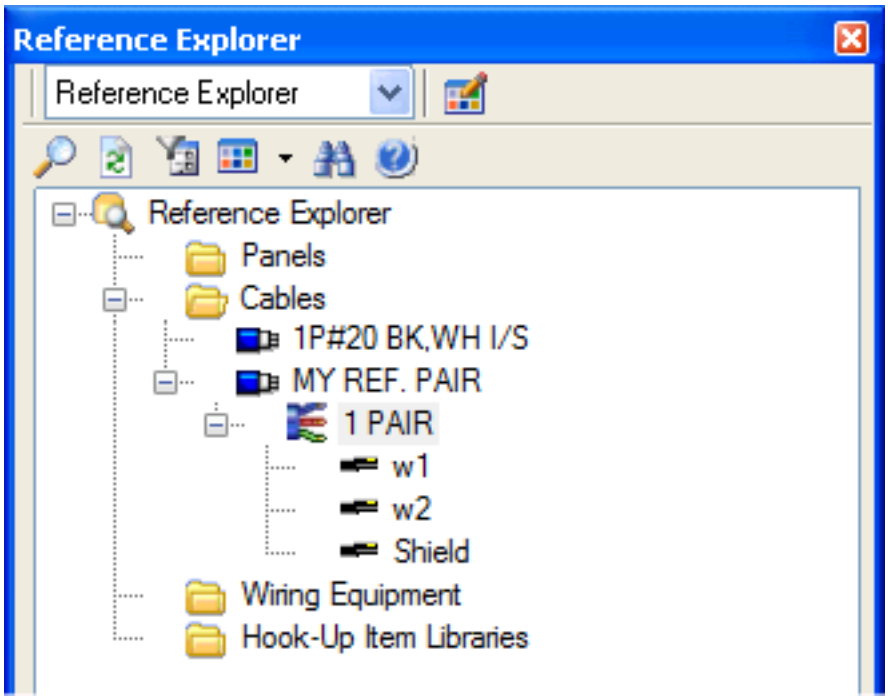
7. When completed, click **Save**.
8. Click **Create** to open the **Cable Properties** dialog box where you can create a new cable.

9. Under **Cable**, type **MY REF. PAIR**. In the **Glands** group box, you can define some gland types by clicking ... next to the **End 1** or **End 2** lists. You can leave all the other properties undefined.



10. Click **OK** to close the **Cable Properties** dialog box.

11. View the cable-set-wire hierarchy in the **Cables** folder of the **Reference Explorer**.



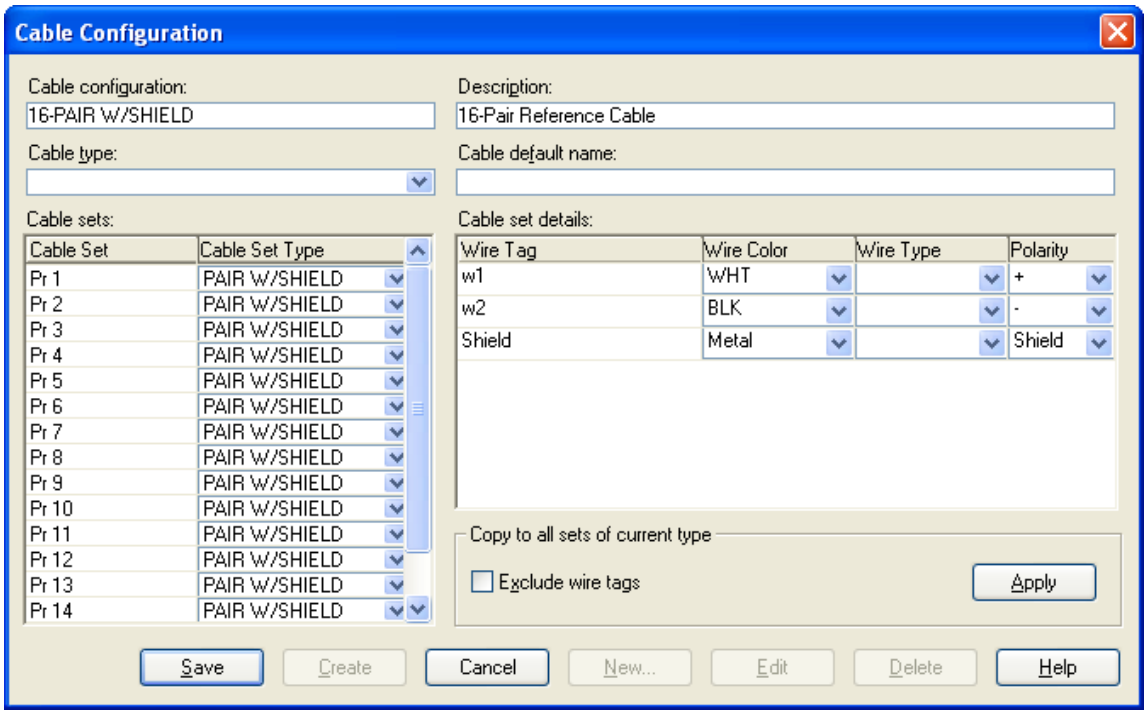
Create a 16-Pair Cable

You will now create a multi-pair cable for your junction box.

1. In the **Reference Explorer**, right-click the **Cables** folder and then on the shortcut menu, click **New** > **Cable**.
2. On the **Cable Configuration** dialog box, click **New** to open the **New Cable Configuration** dialog box.
3. Enter **16** as the total number of sets.
4. Select **PAIR W/SHIELD** from the **Defaultcable set type** list and click **OK** to return to the **Cable Configuration** dialog box.
5. In the **Cable Configuration** field, type **16 PAIR W/SHIELD**.
6. Under **Description**, type **16-Pair Reference Cable**.
7. Under the **Cable Set** column, type **Pr 1** for the first cable set.
8. Continue to number the pairs (**Pr 2**, **Pr 3**, and so forth) for all 16 cable sets.

 TIP You can use the standard Windows shortcut keys **CTRL+C** and **CTRL+V** to save time when creating the values **Pr 1**, **Pr 2**, and so forth.

9. To enter data for every cable set, do the following (as shown):



 a. Select cable set **Pr 1**.
 b. Under the **Cable set wire details** group box, enter the wire tag, the wire color, and the polarity.
 c. Under **Copy to all sets of current type**, clear the **Exclude wire tags** check box to copy the wire tag values for each cable set.
 d. Click **Apply**.

 As shown above, the software copies the wire color, wire type, and polarity data to all the other sets.

10. Click **Save**.
11. Click **Create**.
12. In **Cable Properties** dialog box, under **Cable**, type **16 PAIRS WITH I/S**.
13. Click **OK** to close the **Cable Properties** dialog box.

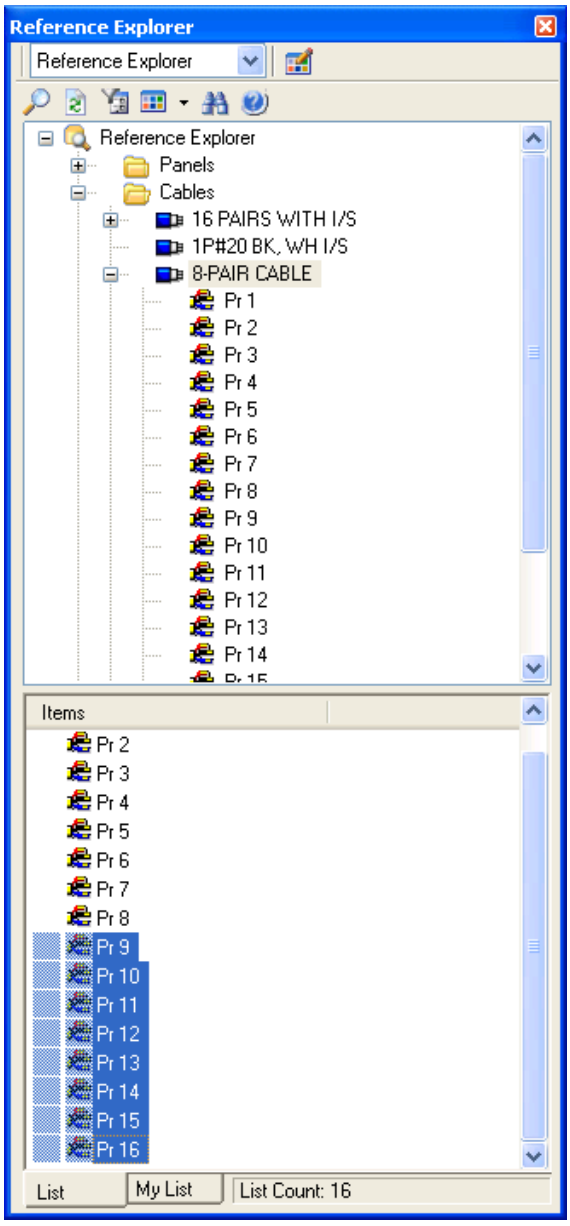
Create a Reference 8-Pair Cable and a 4-Pair Cable for Your DCS Analog Input Card Connection

You will now create these two reference cables by populating the newly created 16-pair cable.

1. In the **Cables** folder of the **Reference Explorer**, right-click cable **16 PAIRS WITH I/S**.
2. Click **Duplicate** on the shortcut menu.
3. On the **Cable Properties** dialog box under **Cable**, type **8-PAIR CABLE**.
4. Leave all the other fields empty and click **OK**.
5. In the **Reference Explorer**, select this newly created cable **8-PAIR CABLE**.

 TIP You can also see all the existing cable sets in the **Items** pane of the **Reference Explorer**.

6. You will now delete the surplus cable sets in cable **8-PAIR CABLE** as follows:
 a. In the **Reference Explorer**, select Cable **8-PAIR CABLE**.
 b. In the **Items** pane, scroll down to pair number **9** (**Pr 9**) and while holding down **Shift**, select pair number **16** (**Pr 16**) to select all the sets between **Pr 9** and **Pr 16**.



7. Press the **Delete** key on your keyboard.
8. When prompted, click **Yes** to confirm the deletion.
9. Duplicate **8-PAIR CABLE** and rename the duplicated cable as **4-PAIR CABLE**.
10. Delete the 4 surplus cable sets for **4-PAIR CABLE**.

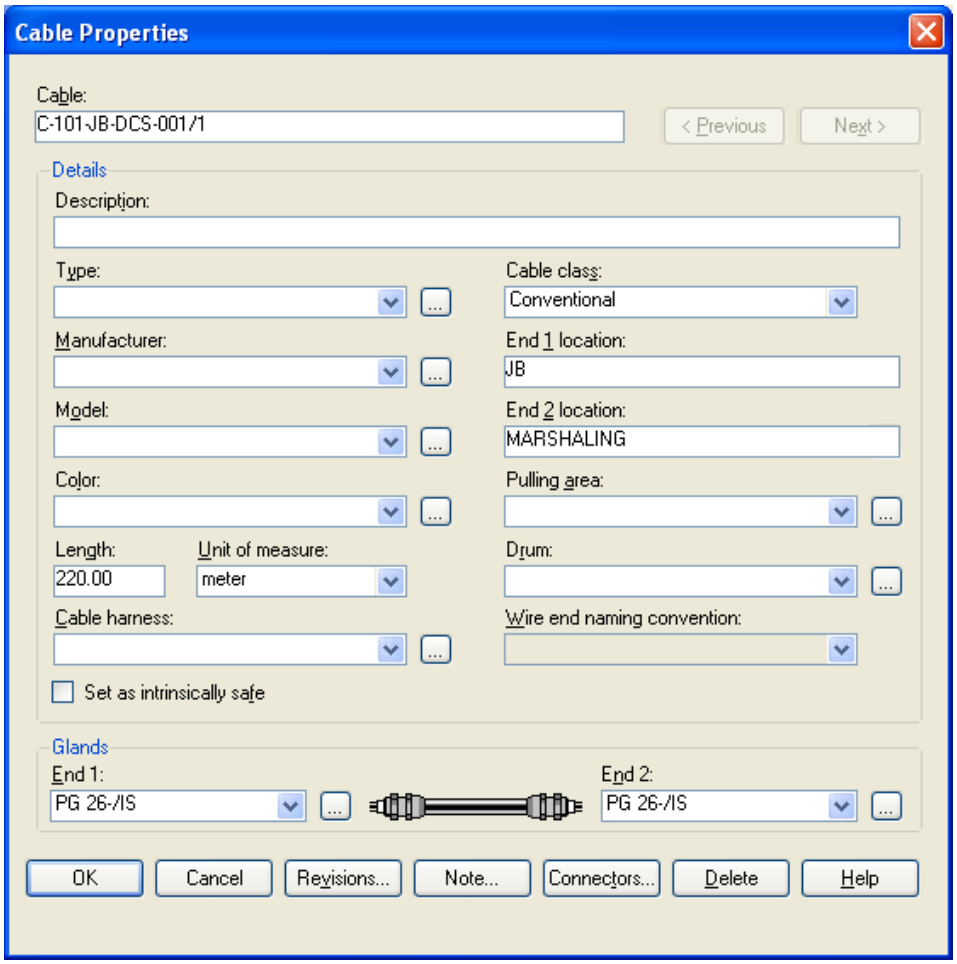
You have now completed the creation of all the reference cables required for this tutorial. You will now copy the reference cables you created to the **Cables** folder in the **Domain Explorer**.

Task 6 — Copying Reference Cables to the Plant

You will now make use of the reference cables in your plant.

Copy Reference Cables with Sets and Wires

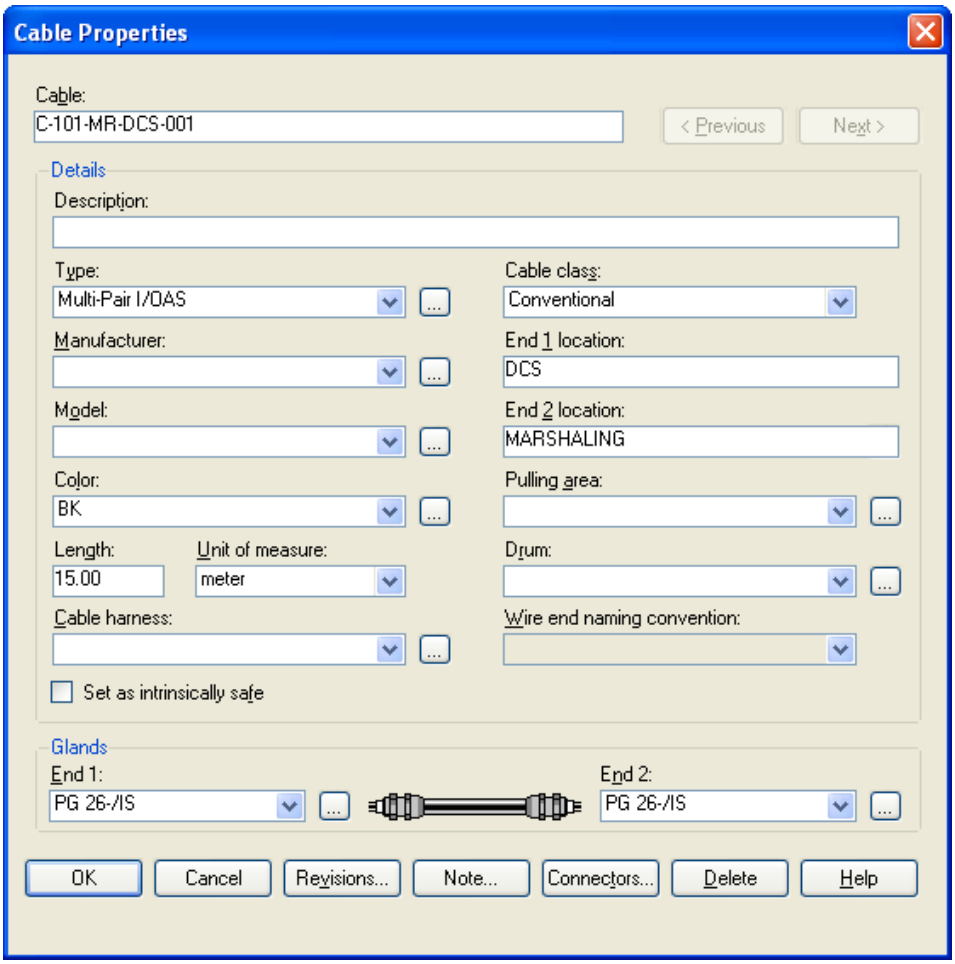
1. Make sure that both the **Domain Explorer** and the **Reference Explorer** are open side by side. (Press **F7** to open the **Domain Explorer** and **F8** to open the **Reference Explorer**.)
2. In the **Reference Explorer**, expand the **Cables** folder.
3. Select cable **16 PAIRS WITH I/S** and drag it to the **Cables** folder in the **Domain Explorer**.
4. On the **Cable Properties** dialog box that opens, enter the cable data as shown.



TIP To select the cable color, cable type or gland and glands, click ... to access the supporting table where you create the required item. This item will then become available in the appropriate list.

5. When done, click **OK**.
6. Define cable **C-101-MR-DCS-001** as follows:
 a. In the **Reference Explorer**, select **8-PAIR CABLE** and drag it to the **Cables** folder in the **Domain Explorer**.

b. On the **Cable Properties** dialog box, enter the cable data as shown.

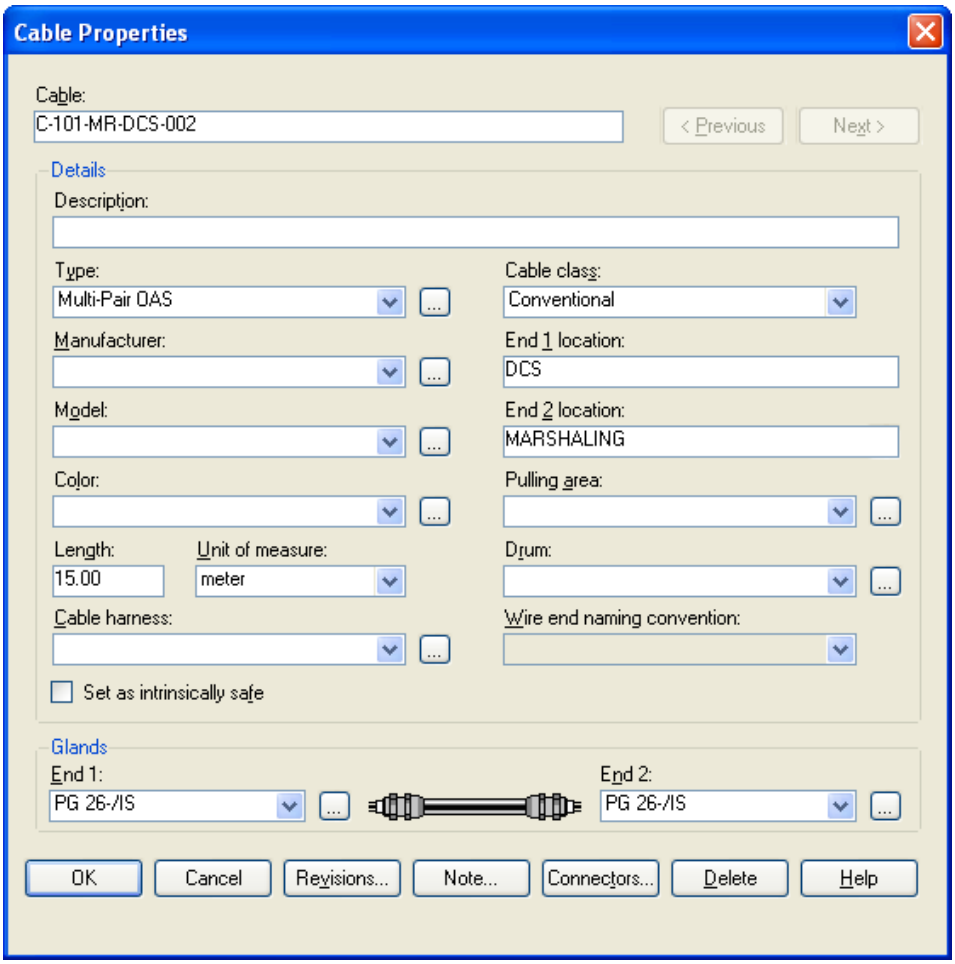


c. Click **OK**.

7. Define **C-101-MR-DCS-002** as follows:

a. In the **Reference Explorer**, select **4-PAIR CABLE** and drag it to the **Cables** folder in the **Domain Explorer**.

b. On the **Cable Properties** dialog box, enter the cable data as shown.



c. Click **OK**.

8. Create and define **C-101-LT-201** as follows:
 a. In the **Reference Explorer**, select **MY REF. PAIR** cable and drag it to the **Cables** folder in the **Domain Explorer**.
 b. On the **Cable Properties** dialog box, change the **Cable** name to **C-101-LT-201**.
 c. Click **OK**.
9. Create and define **C-101-FI-201** as follows:
 a. In the **Reference Explorer**, select **MY REF. PAIR** cable and drag it to the **Cables** folder in the **Domain Explorer**.
 b. On the **Cable Properties** dialog box, change the **Cable** name to **C-101-FI-201**.
 c. Click **OK**.

You have just completed creating all the cables required for this tutorial.

Task 7 — Making Connections

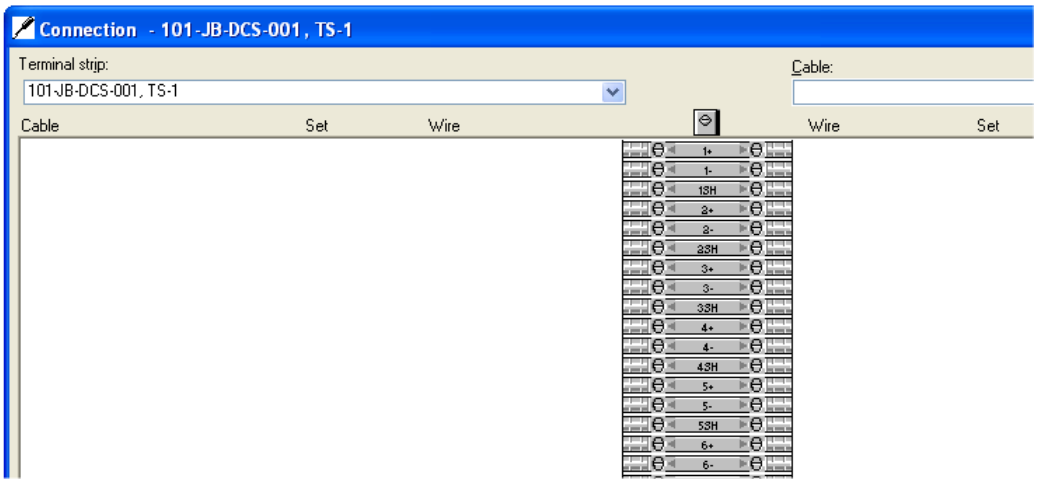
CAUTION Cables cannot be connected to terminal strips before the connection type is defined. Terminal connection type defines how wires are to be connected to a terminal strip, which wire is connected to which terminal, and so forth. Read the **Define a Connection Type** section in the **SmartPlant Instrumentation Detailed Engineering Users Guide** document to learn about the connection type concept and how to create your own connection types. For the purposes of this tutorial, we will use SmartPlant Instrumentation predefined connection types to connect our cables to the panels.

At this stage of the tutorial, you will connect the field wiring (field instruments) to the junction box. You will then connect the junction box to the field side of the marshaling rack. Subsequently, you will assign I/O tags to DCS cards and connect them to the DCS side of the marshaling rack. Once the sides are connected, you will perform an internal cross wiring within the marshaling rack between the field side terminals and the DCS side terminals.

In this set of procedures, you will make the wiring connections for three control loops (**101-F -100**, **101-F -102**, and **101-P -201**). You will make the connections starting from the field devices and working through to the DCS in the control room. In the course of connecting the wires, you will observe how the software handles propagation.

Connect Multi-Pair Cables to Junction Boxes

1. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **Junction Boxes** folder.
2. Right-click **101-JB-DCS-001** and on the shortcut menu, click **Actions** > **Connection**.

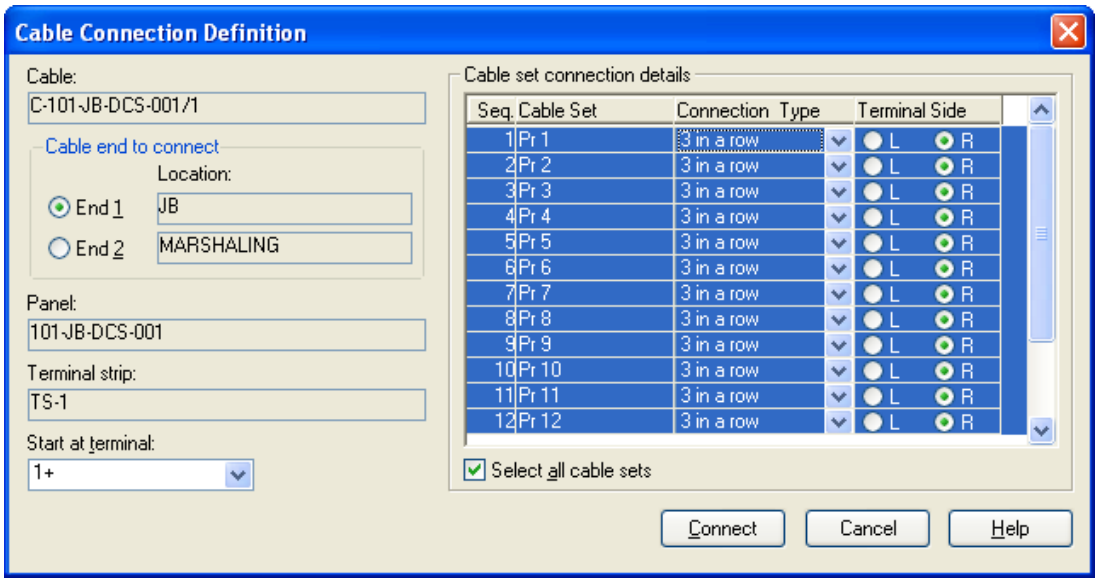


3. Make sure that in the **Terminal strip** list, **101-JB-DCS-001, TS-1** is selected.
4. In the **Domain Explorer**, expand the **Cables** folder and select **C-101-JB-DCS-001/1**.
5. Drag cable **C-101-JB-DCS-001/1** from the **Domain Explorer** to the first terminal at the right side of the terminal strip.

 TIP When you release the mouse button, the **Cable Connection Definition** dialog box opens.
6. Make sure the **End 1 (JB)** option button has been selected (as you are dealing with the junction box end of this cable.)
7. Under **Cable set connection details**, from the **Connection Type** list, select **3 in a row** as the connection type for set **1 (Pr 1)**.

8. If prompted to apply to all cable sets, click **Yes**, or select the **3 in a row** connection type for each set in turn.
9. Verify that the **Select all cable sets** check box is selected.

 The dialog box should now appear as shown:

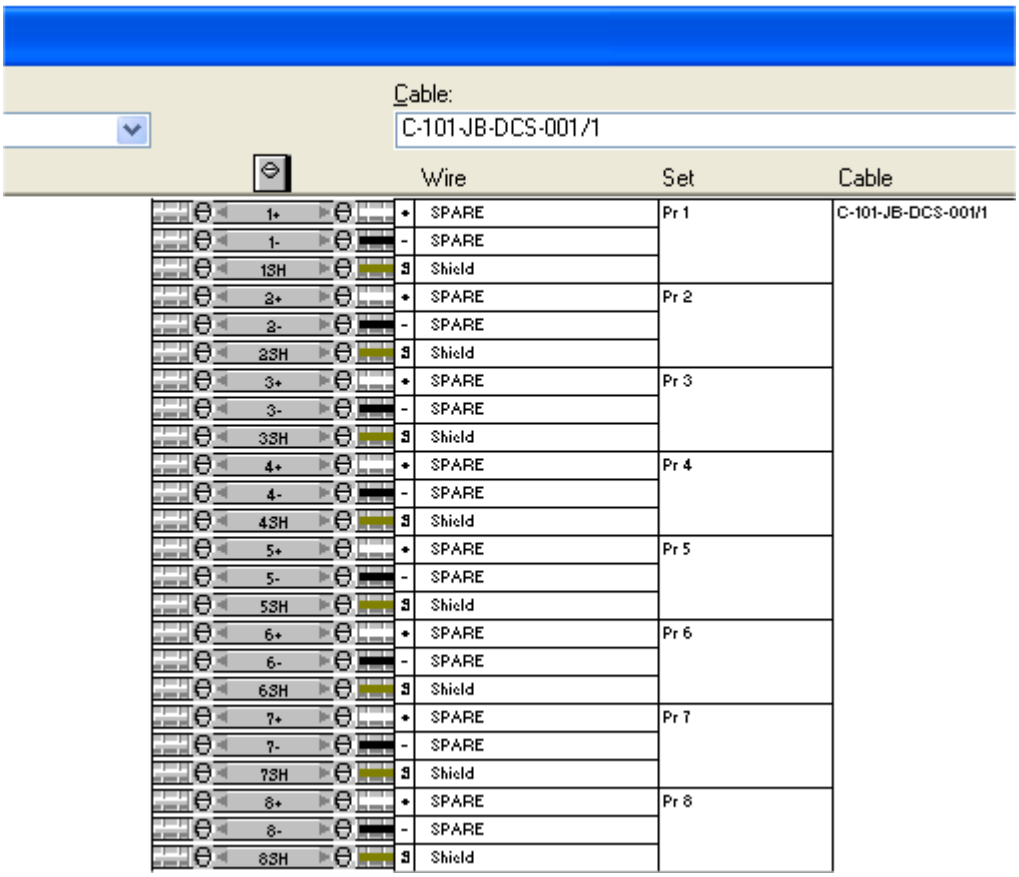


TIPS

- The connection type determines the way in which the individual wires in a cable set are connected, and how many terminals to leave unconnected between the wires in adjacent cable sets.
- For a more comprehensive explanation of connection types, see *SmartPlant Instrumentation User Guide, Wiring Module*.

10. Click **Connect**.

The status bar shows the progress as connections are made. The connections are displayed in the **Connection** window as shown:



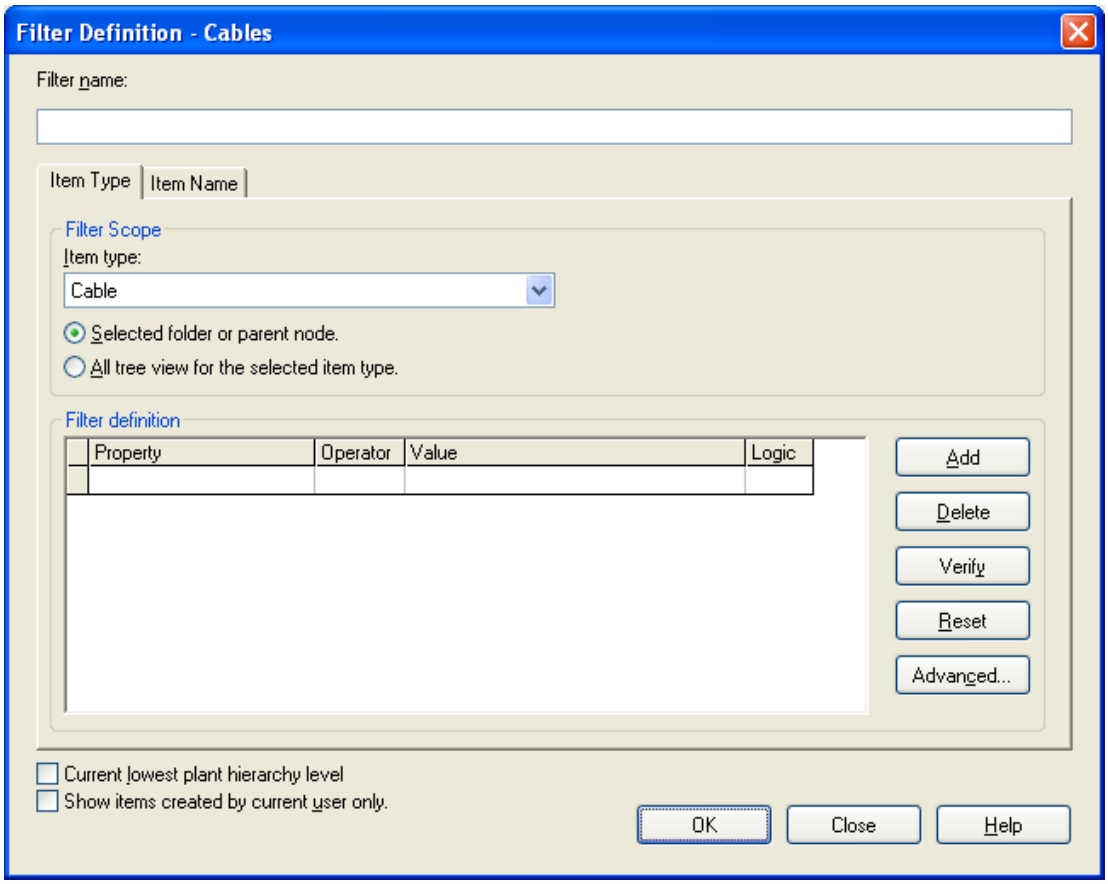
Filter the Cables in the Domain Explorer

In this activity, you will set a filter that allows you to display only the available unconnected field instrument cables.

1. Select the **Cables** folder in the **Domain Explorer**.

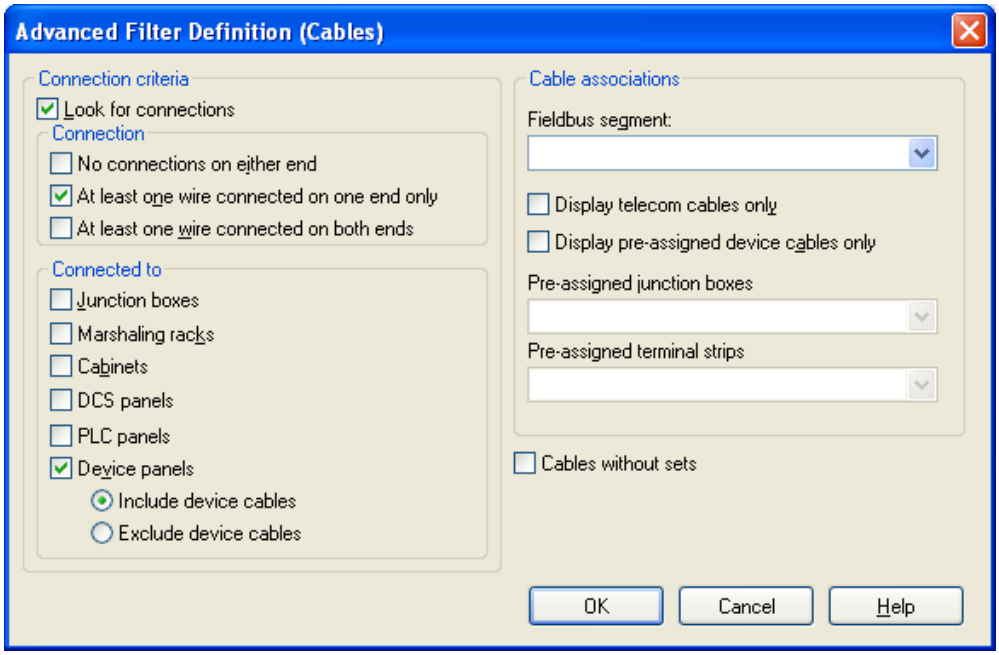
 TIP You can keep the **Connection** window open.

2. Click the filter icon to open the **Filter Definition - Cables** dialog box.



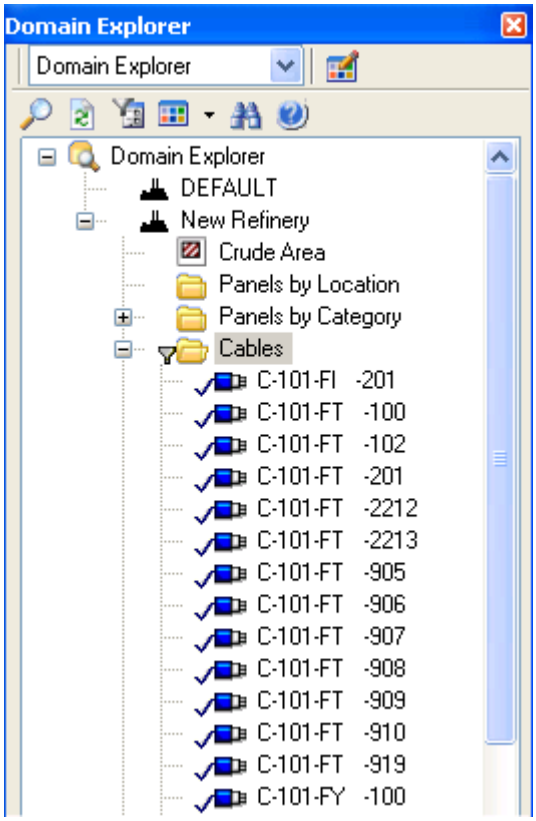
3. Under **Filter name**, type **Temporary Cable Filter** and then click **Advanced** to open the **Advanced Filter Definition – Cables** dialog box.

4. In the **Connection criteria** group box, select the **Look for connections** check box and then set the filter as shown.



5. Click **OK**.
6. Click **OK** on the **Filter Definition - Cables** dialog box and to accept the filter condition.

The **Cables folder** in the **Domain Explorer** is now filtered and the filter indicator now appears next to the **Cables** folder as shown:



7. Right-click the **Cables** folder and then on the shortcut menu, click **Clear Filter**.

Task 8 — Connecting Device Cables

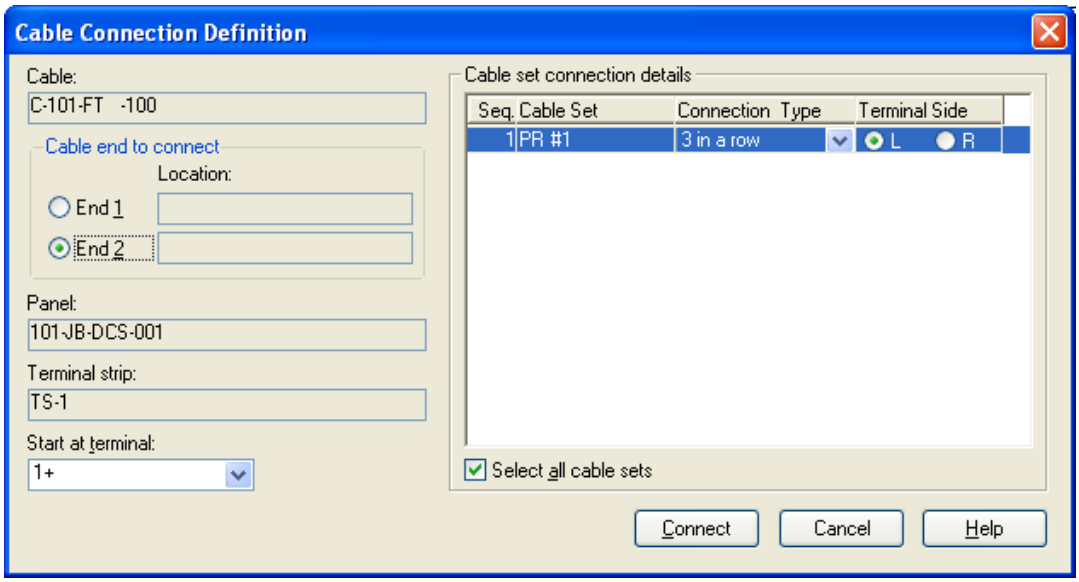
There are two methods for connecting field device cables, both of which you will learn:

- Individual connection (one by one).
- Batch connection.

Connect a Single Field Device to a Junction Box

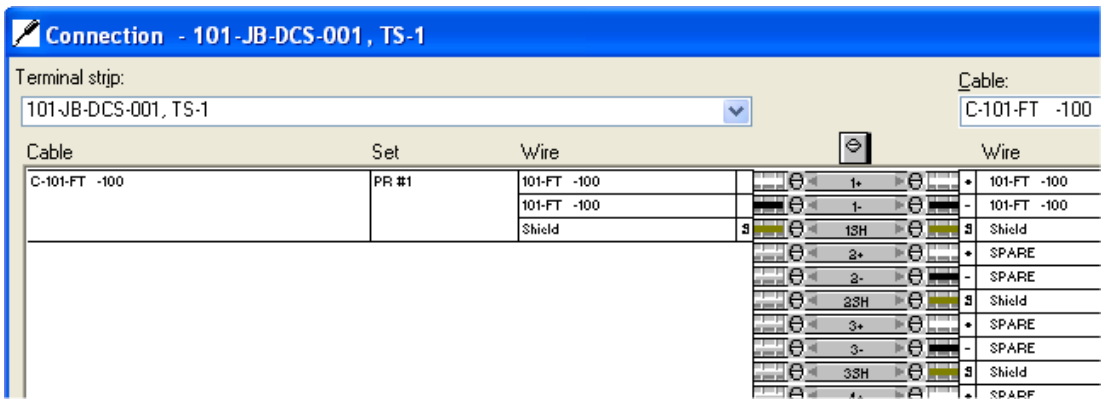
1. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **Junction Boxes** folder.
2. Right-click **101-JB-DCS-001** and on the shortcut menu, click **Actions** > **Connection**.
3. With the **Connection** window open, select cable **C-101-FT-100** in the **Cables** folder of the **Domain Explorer**. Drag the **Domain Explorer** to the right side of the window for better viewing.
4. Drag cable **C-101-FT-100** from the **Domain Explorer** to the left side of terminal **1+** in the **Connection** window and drop it there.
5. On the **Cable Connection Definition** dialog box, select **End 2** (the JB side) for connection, select the **Select all cable sets** check box and use **3 in a row** as the connection type.

The dialog box should appear as shown:



6. Click **Connect** to make the connection.

 After the connection has been made, the following should appear in the Connection window:



NOTE Wires for cable **C-101-FT-100** labeled **SPARE** before connection of the field device take the names of the tag numbers of the instruments that have been defined as field devices. This is an example of tag number signal.

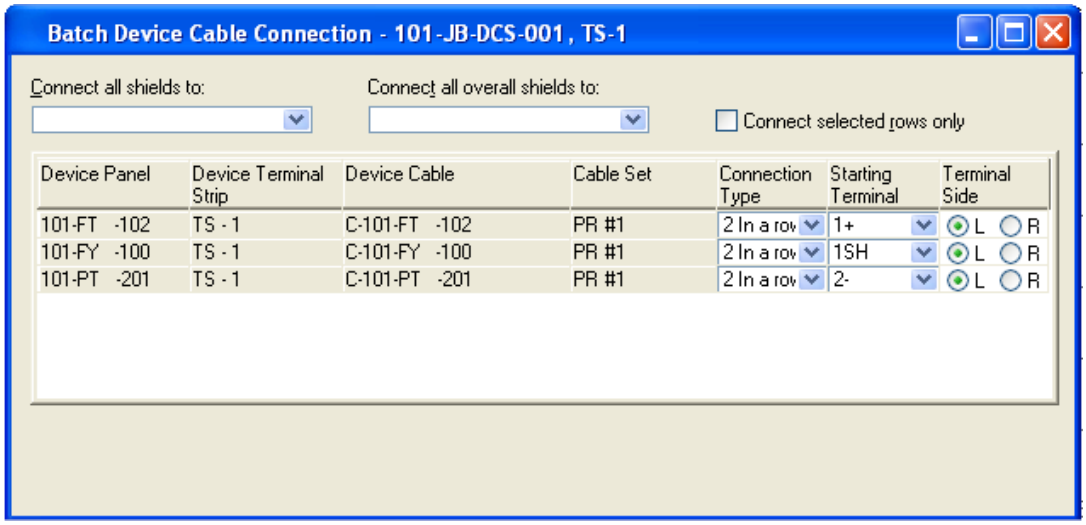
Connect Field Instrument Cables in Batch Mode

In the previous exercise, you learned how to connect a single cable to a junction box. Now we will show you how to connect multiple device cables in batch mode. Let's connect all the remaining "single" cables to the same junction box.

1. With the **Connection – 101-JB-DCS-001, TS-1** window open, on the **Actions** menu, click **Batch Connection** to open the **Batch Device Cable Connection** window.
2. In the **Domain Explorer**, select the **Cables** folder.
3. In the **Items** pane of the **Domain Explorer**, select Cable **C-101-FT-102**, hold down **Ctrl** then scroll down the device cable list and select cables **C-101-FY-100** and **C-101-PT-201**.

 NOTE You can include any other cables that you have.

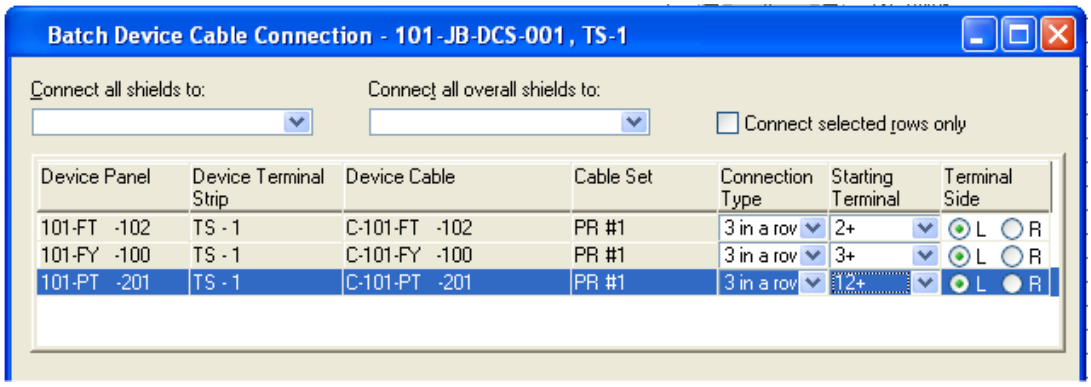
4. Drag the selected device cables from the **Items** pane of the **Domain Explorer** to the data window of **Batch Device Cable Connections**.



5. For cable **C-101-FT-102**, in the **Connection Type** column, select **3 in a row**.
6. Click **Yes** if prompted to apply the connection type to all sets except for shields.
7. Make sure that the following values appear in the **Starting Terminal** column.

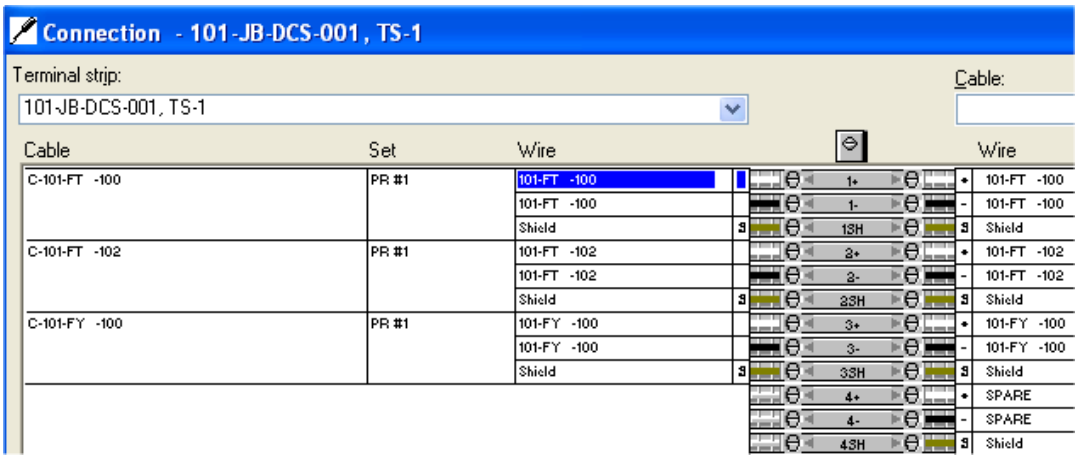
Device Cable	**Starting Terminal**
C-101-FT-102	2+
C-101-FY-100	3+
C-101-PT-201	12+

The window should now appear as shown:



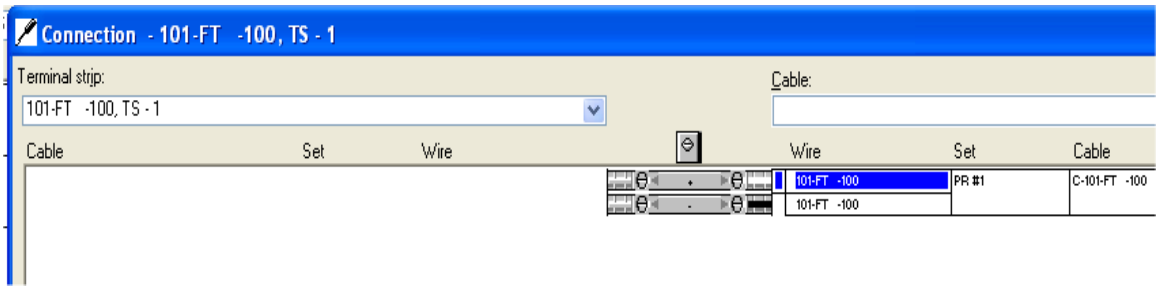
8. Click **Actions** > **Connect** to connect the selected device cables in batch mode.
9. Minimize the **Batch Device Cable Connections** window.
10. In the **Connection** window, click  on the module toolbar and observe that the tag number signals have been propagated to the right side of the terminals.

11. In the **Connection** window, select the first wire on the left side of **101-FT -100** as shown.



12. Click on the module toolbar to display the other side of the single cable.

 The data display changes as follows:

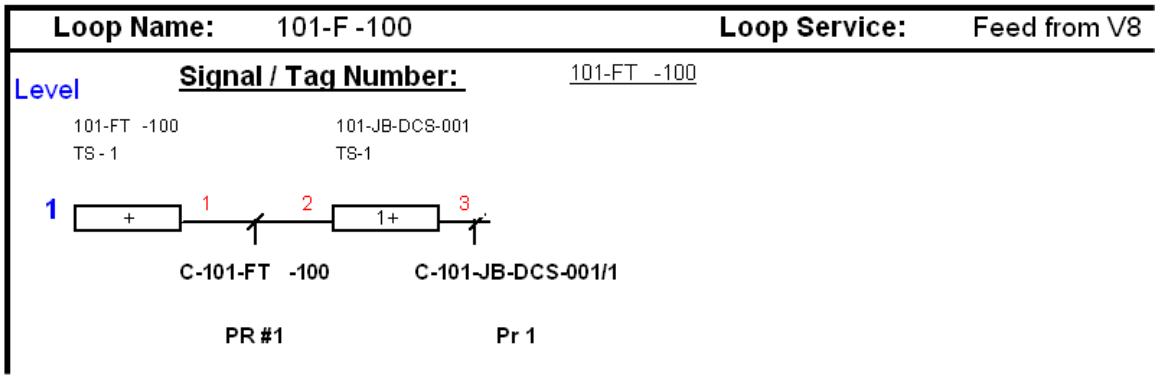


13. Click again to return to the junction box connections.
14. With the first wire on the left side of **101-FT -100** still selected, click to display the **Point-to-Point Wiring Diagram**.
15. When prompted to preview the report, click **Yes**.

 TIP If the diagram script is too small and hard to read, increase the font size or zoom the image as described in the following steps.
16. To increase the font size:
 a. Click **View** > **Font Size**.
 b. Use the spinner to set the font size to 7 (seven).
 c. Click **OK**.

 TIP Try experimenting with different font sizes to see which suits you best.
17. To zoom the image:
 a. On the module toolbar, click .
 b. Click **Custom**, and in the adjacent field, type in 140%.
 c. Click **OK**.

The image should appear similar to what is shown:



18. Click to print out the diagram if needed.
19. Click to close the Point-to-Point Wiring Diagram.
20. Click to close each **Connection** window.

Connect Field Side Wiring to a Marshaling Rack

1. In the **Domain Explorer**, under **Panels By Category** > **Marshaling Racks**, right-click **101-MR-DCS-001**, and on the shortcut menu, click **Actions** > **Connection**.
2. Ensure that in the **Terminal strip** list, terminal strip **101-MR-DCS-001, FT-1** is selected.
3. Drag cable **C-101-JB-DCS-001/1** from the **Cables folder** in the **Domain Explorer** to the first terminal at the left side of the terminal strip.
4. On the **Cable Connection Definition** dialog box, select **End 1** (JB) to select the junction box side for connection.

 NOTE All sets on cable end 1 are already connected, as indicated by (screw heads) in the **Side** column.
5. Click the **End 2** (MARSHALING) option button to select the marshaling rack side.
6. Make sure that 1+ is selected for **Start at terminal** list, and the **Select all cable sets** check box is selected.
7. Make sure that the connection type is 3 in a row for all sets.
8. Click **Connect**.

 Observe that the tag numbers have been propagated up to the left side of the terminals.

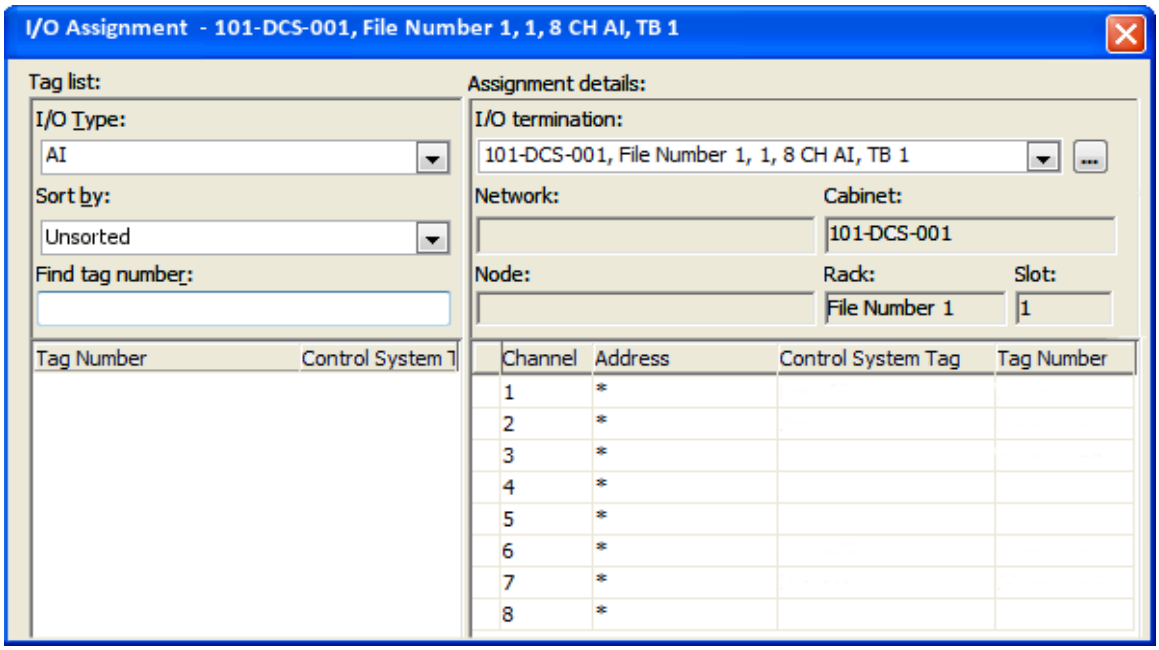
 TIP Open the Point-to-Point Diagram again for the first wire named **C-101-FT-100**. Notice the progress.
9. Click to close the **Connection** window.

Task 9 — DCS Management

Now that you have completed the required field wiring you are going to work on the DCS side. You will perform I/O assignment, connect the I/O card to the marshaling rack, and connect the input and output analog cables.

Assign I/Os to the DCS

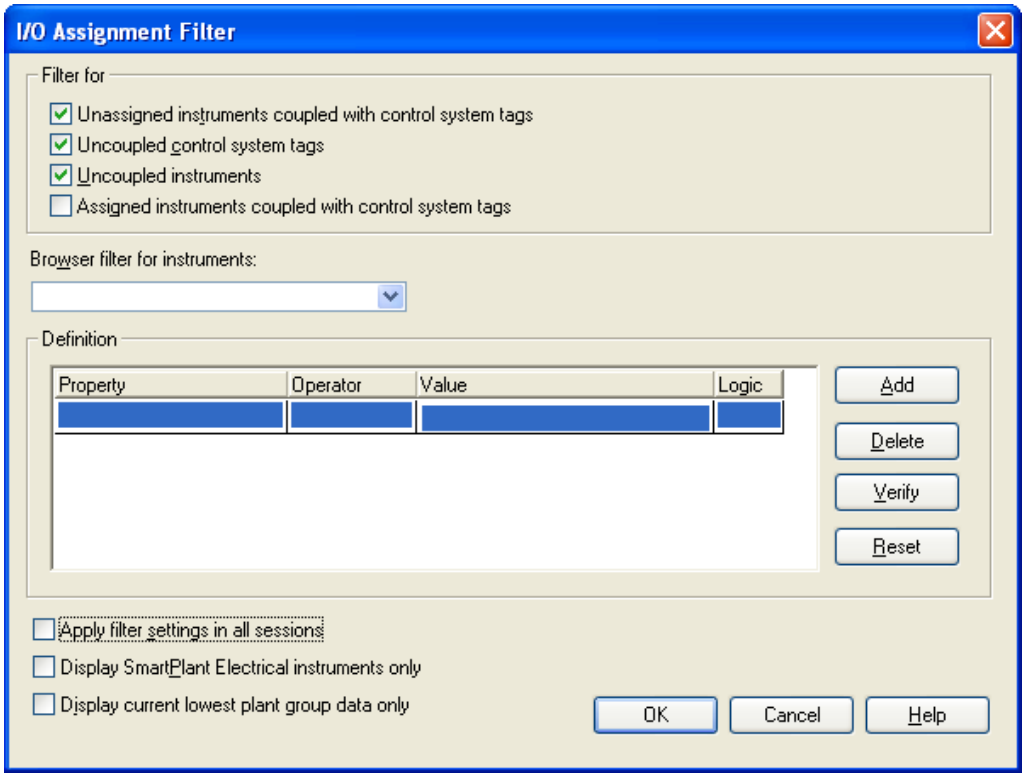
1. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **DCS Panels** folder.
2. Right-click **101-DCS-001**, then on the shortcut menu, click **Actions** > **I/O Assignment**.



TIP If the **I/O Assignment Type** dialog box opens, select the **I/O assignment** option and click **OK**.

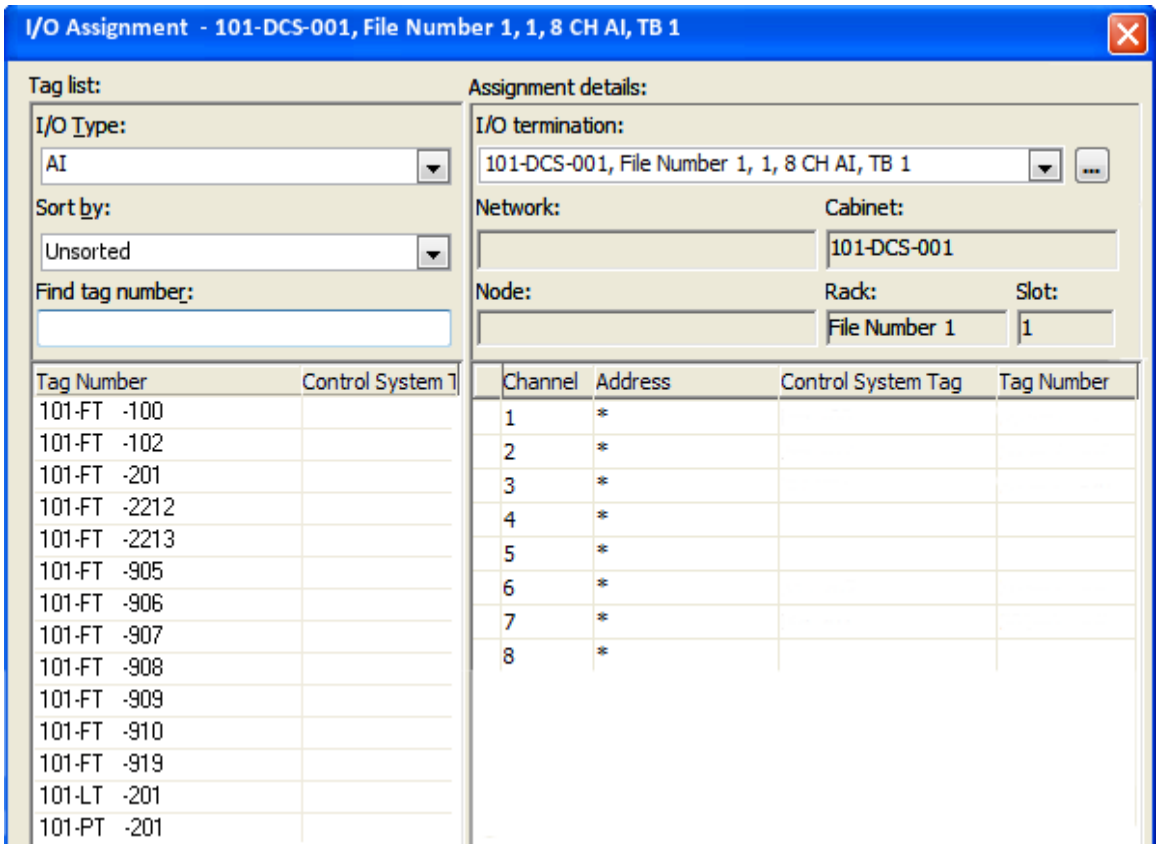
3. Make sure that in the **Assignment details** pane, under **I/O termination**, the software displays **101-DCS-001, File Number 1, 1, 8 CH AI, TB1**.

4. On the module toolbar, click  to open the **I/O Assignment Filter** dialog box, define the filter as shown below, and click **OK**.



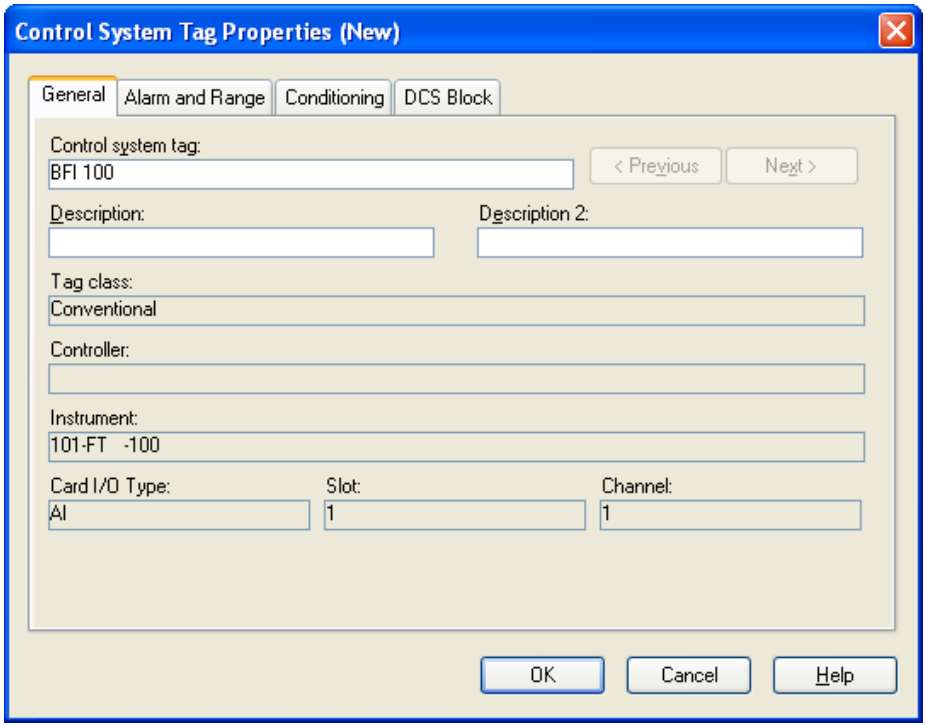
5. In the **Tag list** pane of the **I/O Assignment** window, from the **Sort by** list, select **Tag number** to sort the display of tags in the pane.

The **I/O Assignment** window should now appear as shown:



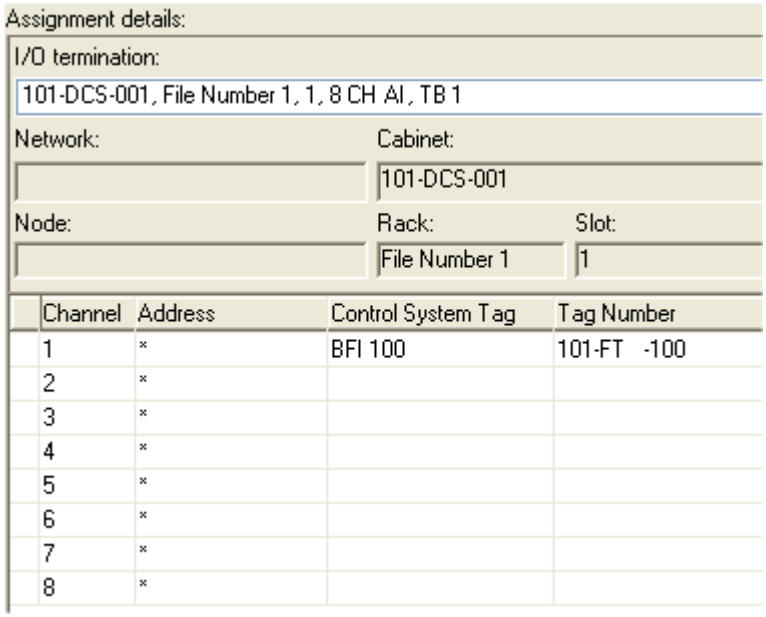
6. Select tag **101-FT-100** and drag it from the **Tag list** to **Channel 1** in the **Assignment details** pane.

7. On the **Control System Tag Properties** dialog box, click in the **Control system tag** box, and change the name of the control system tag to **BFI 100**.



8. Click **OK** to return to the **I/O Assignment** window.

 The I/O assignment of tag 101-FT -100 appears as shown:

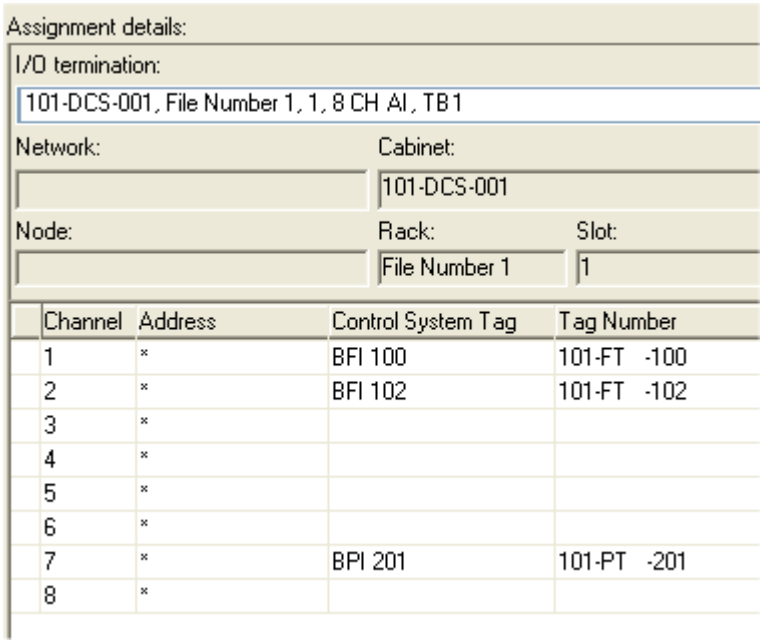


Channel	Address	Control System Tag	Tag Number
1	*	BFI 100	101-FT -100
2	*		
3	*		
4	*		
5	*		
6	*		
7	*		
8	*		

9. Repeat the above steps to do the following:
 a. Select tag **101-FT -102**, assign it to channel **2**, and name its coupled control system tag **BFI 102**.

b. Select Tag **101-PT -201**, assign it to channel **7**, and name its coupled control system tag **BPI 201**.

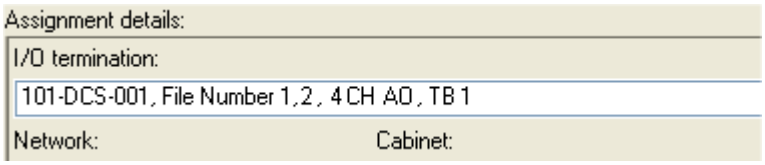
The assignments should now appear as follows:



Effect I/O Assignment for the Analog Tag 101-FY-100

Now we will do the I/O assignment for the analog output tag 101-FY -100. Do the following in the **I/O Assignment** window:

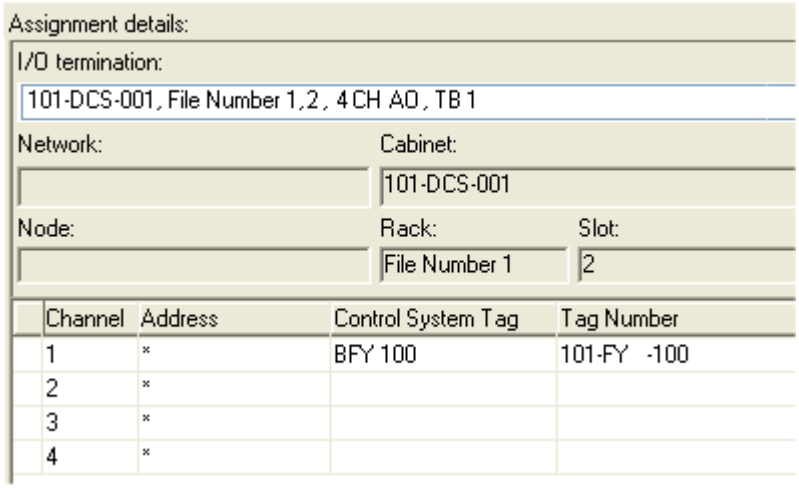
1. In the **Assignment details** pane, from the **I/O termination** list, select the following.



Note that tag number **I/O type** has changed to **AO**.

2. Assign tag **101-FY -100** to channel **1**.

3. Name its coupled CS tag **BFY 100** and click **OK**.

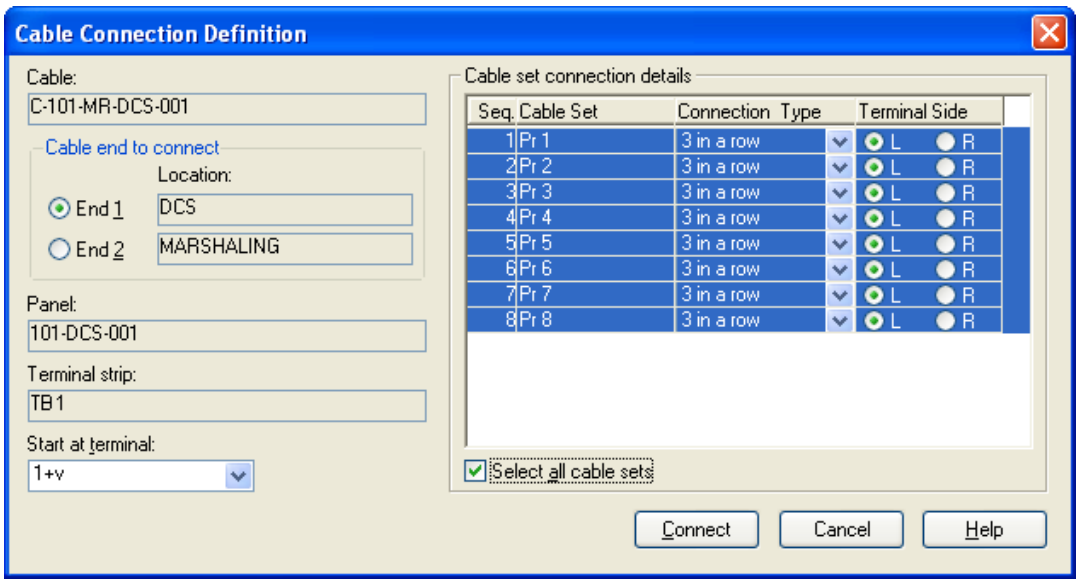


4. Click  to close the **I/O Assignment** window.

You have just completed the I/O assignment of both analog input and output cards. Now you will connect the I/O cards to the marshaling rack, and connect the input and output analog cables.

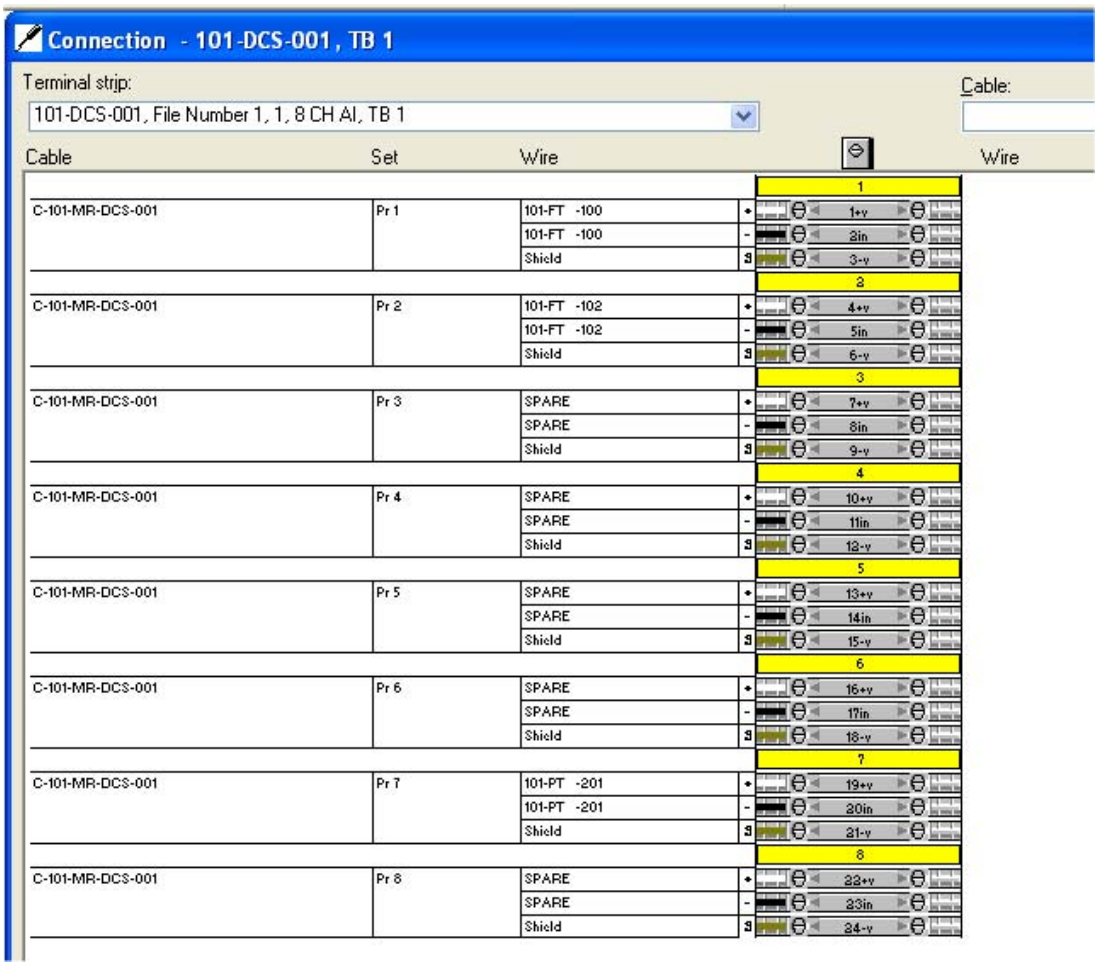
Connect the Analog Input Cable to the DCS Side

1. In the **Domain Explorer**, right-click the **101-DCS-001** DCS panel, and on the shortcut menu, click **Actions** > **Connection**.
2. Make sure that in the **Terminal strip** list, the following terminal and its parent hierarchy is selected: **101-DCS-001, File Number 1, 1, 8 CH AI, TB 1**.
3. In the **Cables** folder of **Domain Explorer**, select cable **C-101-MR-DCS-001** and drag it to the left side of terminal **1+v**.
4. On the **Cable Connection Definition** dialog box, click the **End 1** (DCS) option button and define the other options as shown.



5. Click **Connect** to make the connection.

The **Connection** window now reopens displaying the cable connection to the DCS:



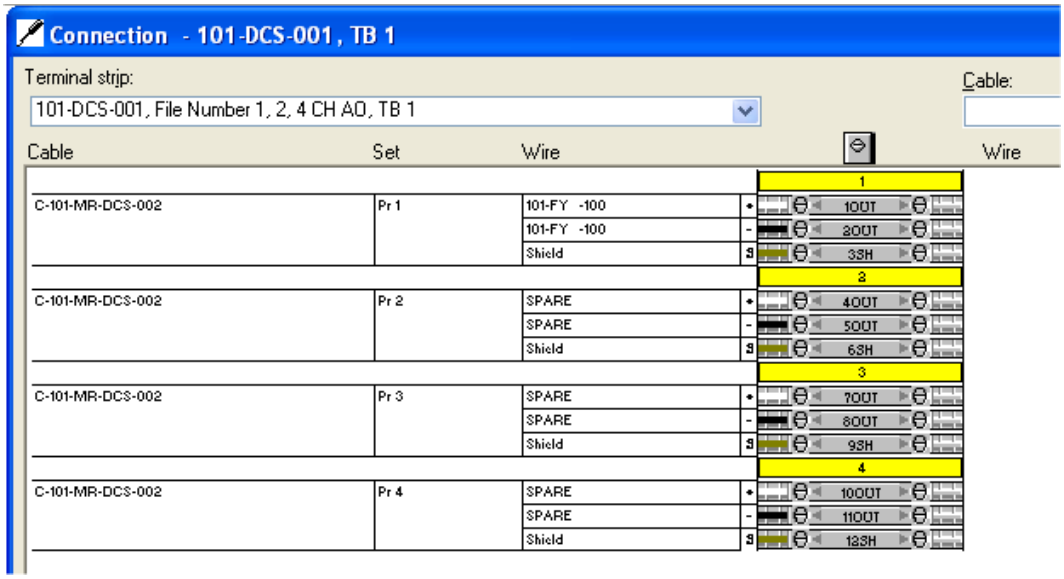
TIP The wires for sets **Pr 1**, **Pr 2**, and **Pr 7** include tag numbers because you have already made I/O assignments for these tags. Scroll down to **Pr 7** and view the wire names.

6. Keep the **Connection** window open.

Connect the Analog Output Cable to the DCS Side

1. In the **Connection** window, from the **Terminal strip** list, select strip **101-DCS-001, File Number 1, 2, 4 CH AO TB 1**.
2. In the **Cables** folder of **Domain Explorer**, select cable **C-101-MR-DCS-002** and drag it to the left side of terminal **+1OUT**.
3. On the **Cable Connection Definition** dialog box, select the **End 1** (DCS) option button to connect the first side of the cable.
4. Click the **Select all cable sets** check box and select **3 in a row** as the connection type for all the sets.
5. Click **Connect** to make the connection.

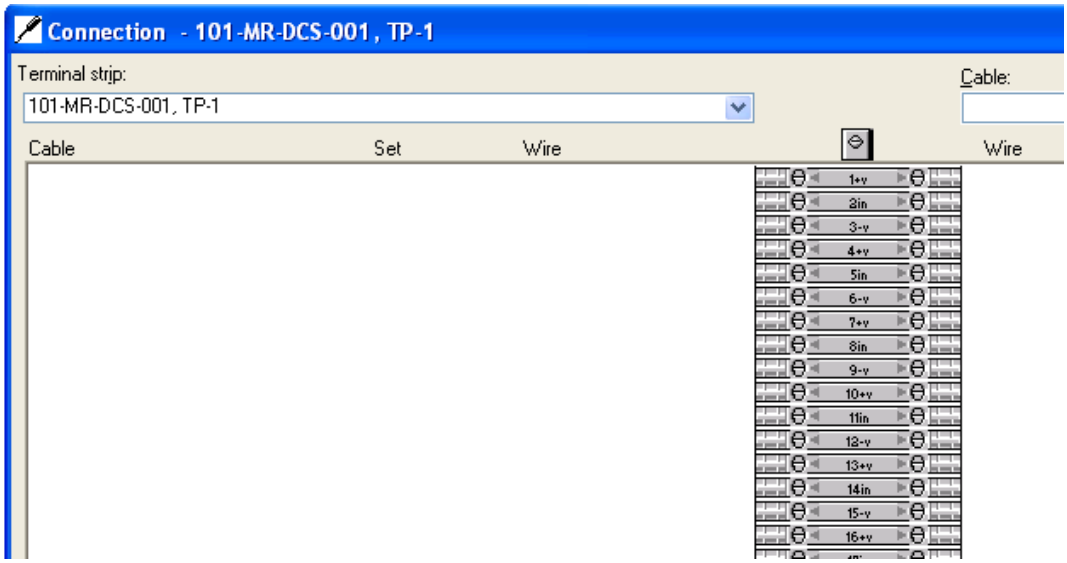
The **Connection** window now reopens where you can see the **101-FY -100** signal propagation in accordance with the I/O assignment you previously made:



6. Click  to close the **Connection** window.

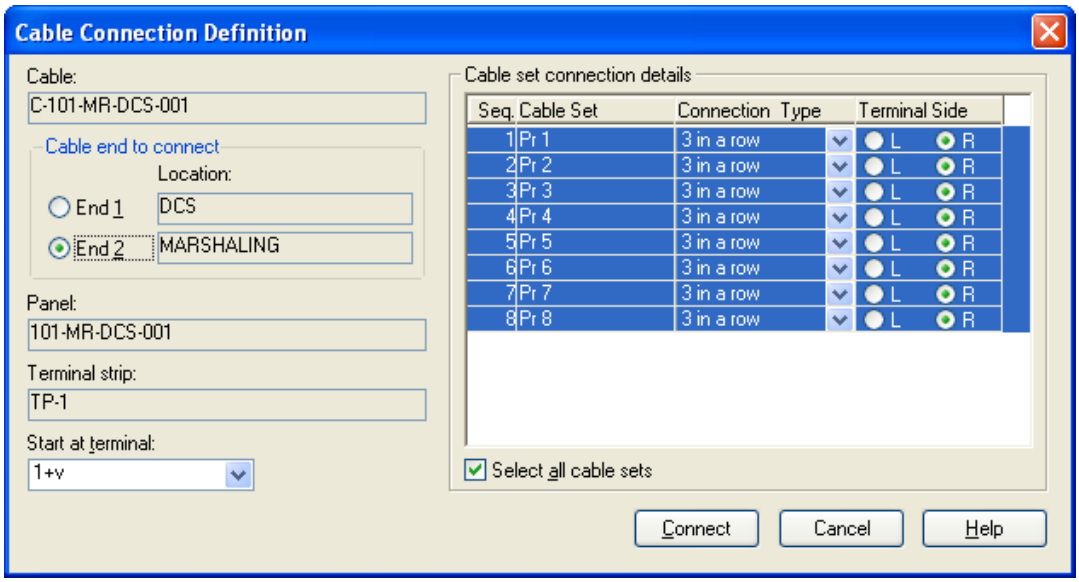
Connect the Analog Input and Output Cables to the Marshaling Rack Side

1. In the **Domain Explorer**, expand the **Marshaling Racks** folder and right-click 1**01-MR-DCS-001**, then on the shortcut menu, click **Actions** > **Connection**.
2. In the **Connection** window, from the **Terminal strip** list, select strip **101-MR-DCS-001, TP-1**.



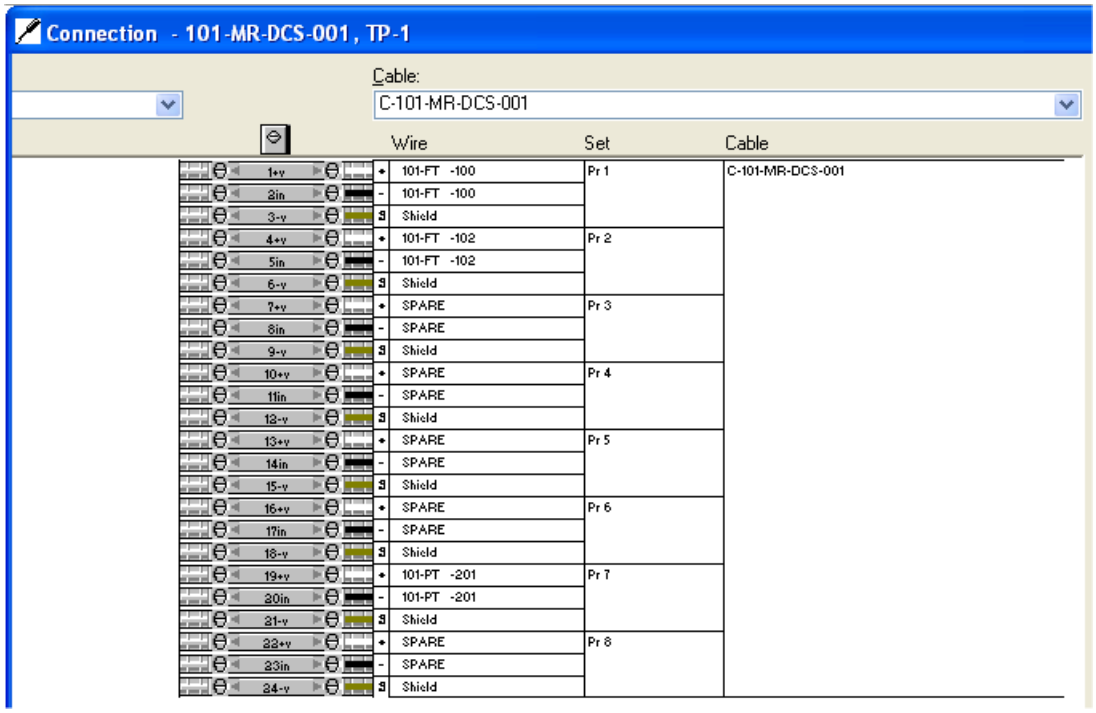
3. In the **Cables** folder of **Domain Explorer**, select cable **C-101-MR-DCS-001** and drag it to the right side of terminal **1+v**.

4. On the **Cable Connection Definition** dialog box, make your selections as shown.



5. Click **Connect** to make the connection.

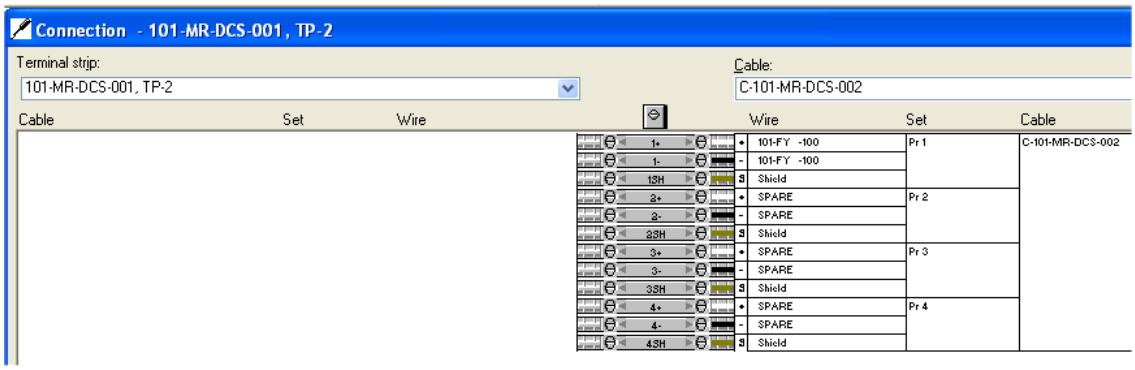
 The **Connection** window now opens where you can see the signal propagation. Use the horizontal scroll bar to see the wire, set, and cable values on the right side of the terminals:



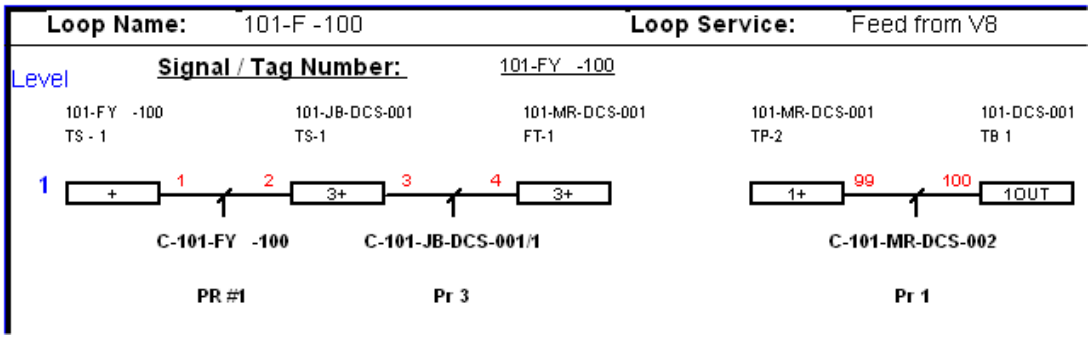
6. In the **Connection** window, from the **Terminal strip** list, select **101-MR-DCS-001, TP-2**.
7. In the **Cables** folder of **Domain Explorer**, select cable **C-101-MR-DCS-002** and drag it to the right side of terminal **1+**.
8. On the **Cable Connection Definition** dialog box, select the **End 2** (MARSHALING) option button to connect the second side of the cable.

9. Click the **Select all cable sets** check box and select **3 in a row** as the connection type for all the sets.
10. Click **Connect**.

 The **Connection** window should now display the propagated tag signal as follows. Use the horizontal scroll bar to see the wire, set, and cable values on the right side of the terminals:



11. In the **Connection** window, click wire **101-FY-100** next to terminal **1+** and then click to open the Point-to-Point Wiring Diagram.



 Note the wiring discontinuity at the marshaling rack, as the required wiring has not yet been completed

12. Print out the Point-to-Point Wiring Diagram if needed and then click to close it.
13. Click to close the **Connection** window.

Task 10 — Cross Wiring the Signals in the Marshaling Rack

As you have seen, there is wiring on both sides of marshaling rack 101-MR-DCS 001. On the field (left) side of the marshaling rack, terminal strip FT-1 is connected to junction box 101-JB-DCS-001. The signals in this wiring segment are 101-FT -100, 101-FT -102, 101-FY -100, and 101-PT -201.

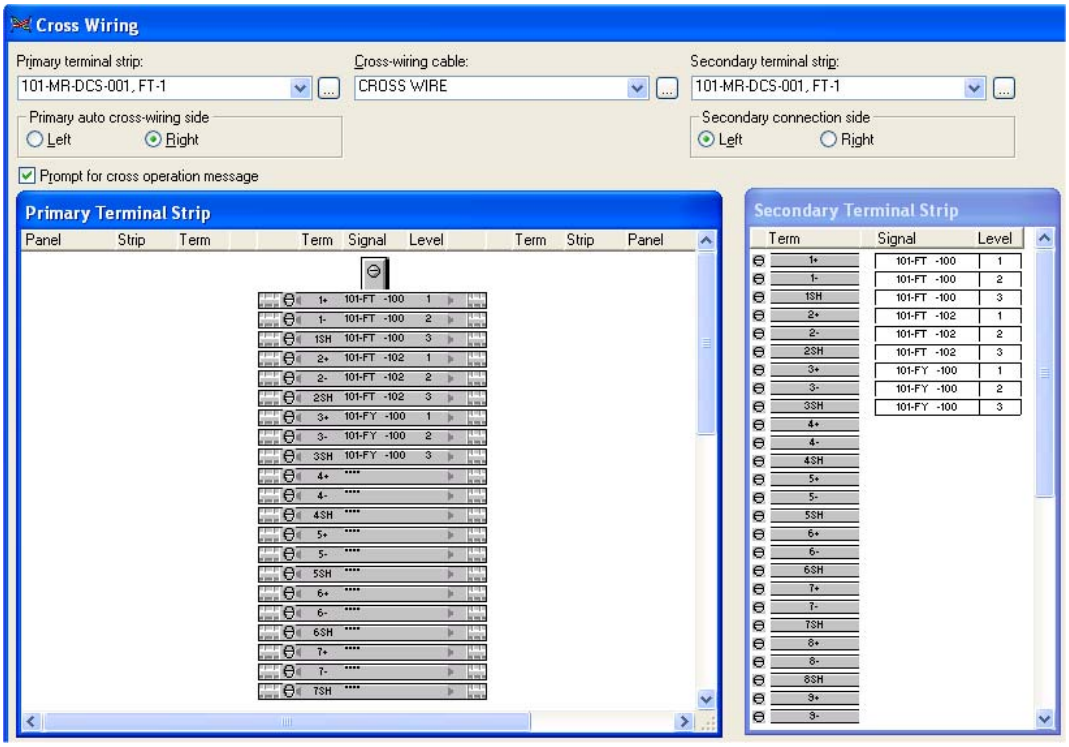
On the DCS (right) side of the marshaling rack, there are two terminal strips — one terminal strip (TP-1) for the analog inputs to the DCS of tags 101-FT -100, 101-FT- 102, and 101-PT -201, and the other terminal strip (TP-2) for the analog output FY-100.

The missing link is the cross wiring between these three terminal strips. You will now connect them using the SmartPlant Instrumentation cross-wiring feature.

Cross Wire One Wire Manually

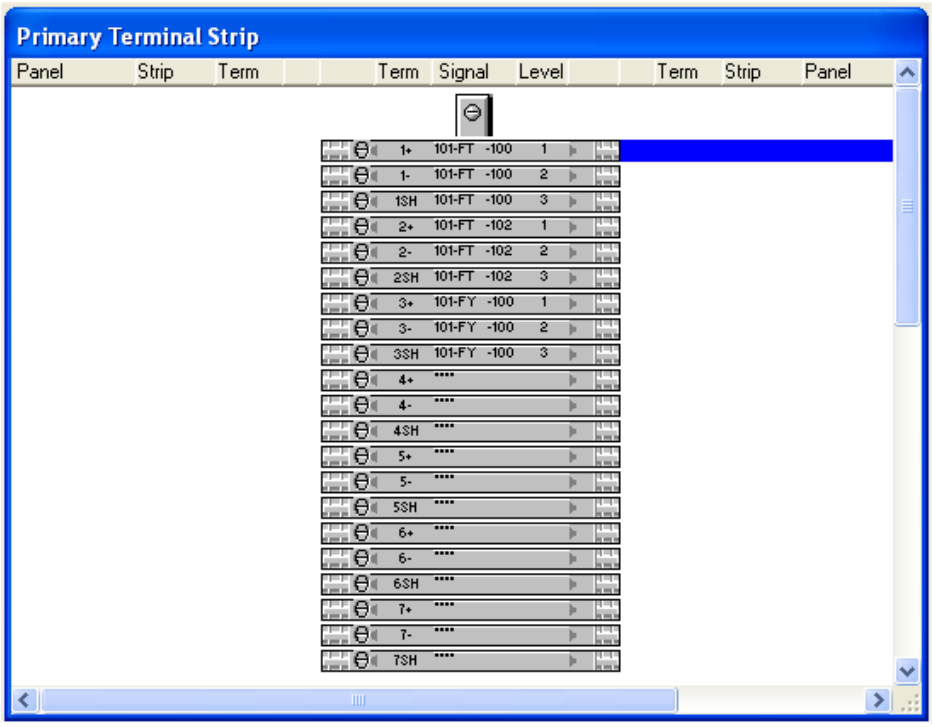
1. In the **Domain Explorer**, expand the **Panels by Category** > **Marshaling Racks** hierarchy and then select marshaling rack **101-MR-DCS-001**.
2. Right-click the selected marshaling rack and then on the shortcut menu, click **Actions** > **Cross Wiring**.

 The **Cross Wiring** window opens:



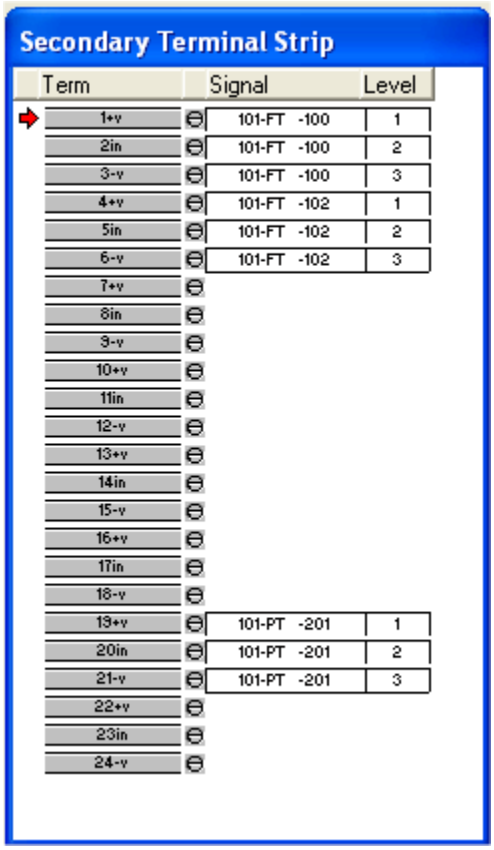
3. Under **Primary terminal strip,** make sure that terminal strip **101-MR-DCS-001**, **FT-1** is selected.
4. In the **Primary auto cross-wiring side** group box, select the **Right** option.

5. In the **Primary Terminal Strip** pane, select the first row (to the right of signal **101-FT-100**, terminal **1+**, level **1**).

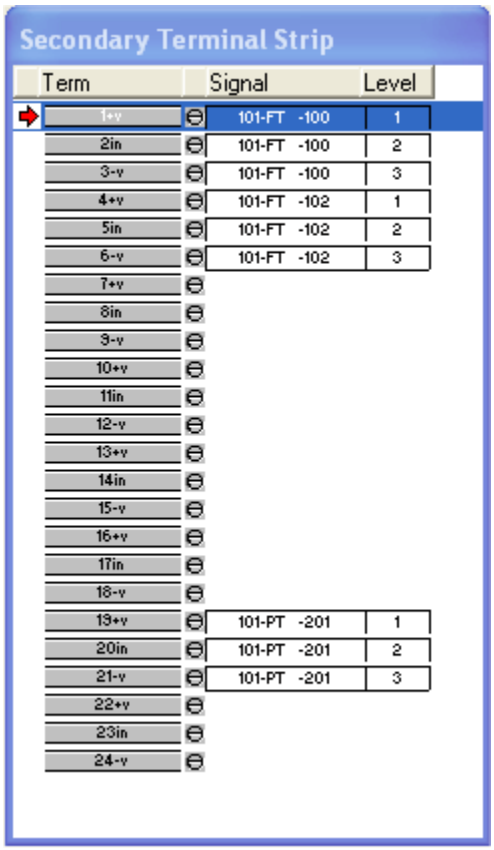


6. On the module toolbar, click the **Target** icon to find matching terminals and display them in the **Secondary Terminal Strip** pane.

The matching terminal is indicated by the icon:



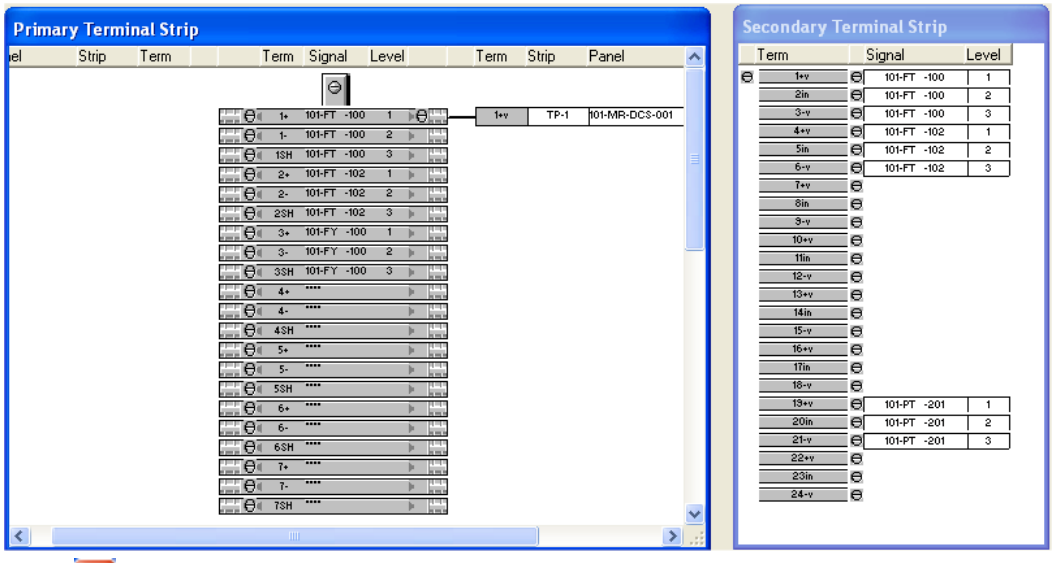
7. Click the matching terminal in the **Secondary Terminal Strip** pane to select it.



A prompt appears, requesting your confirmation to make the cross-wiring connection.

8. Click **Yes** to confirm the cross wiring.

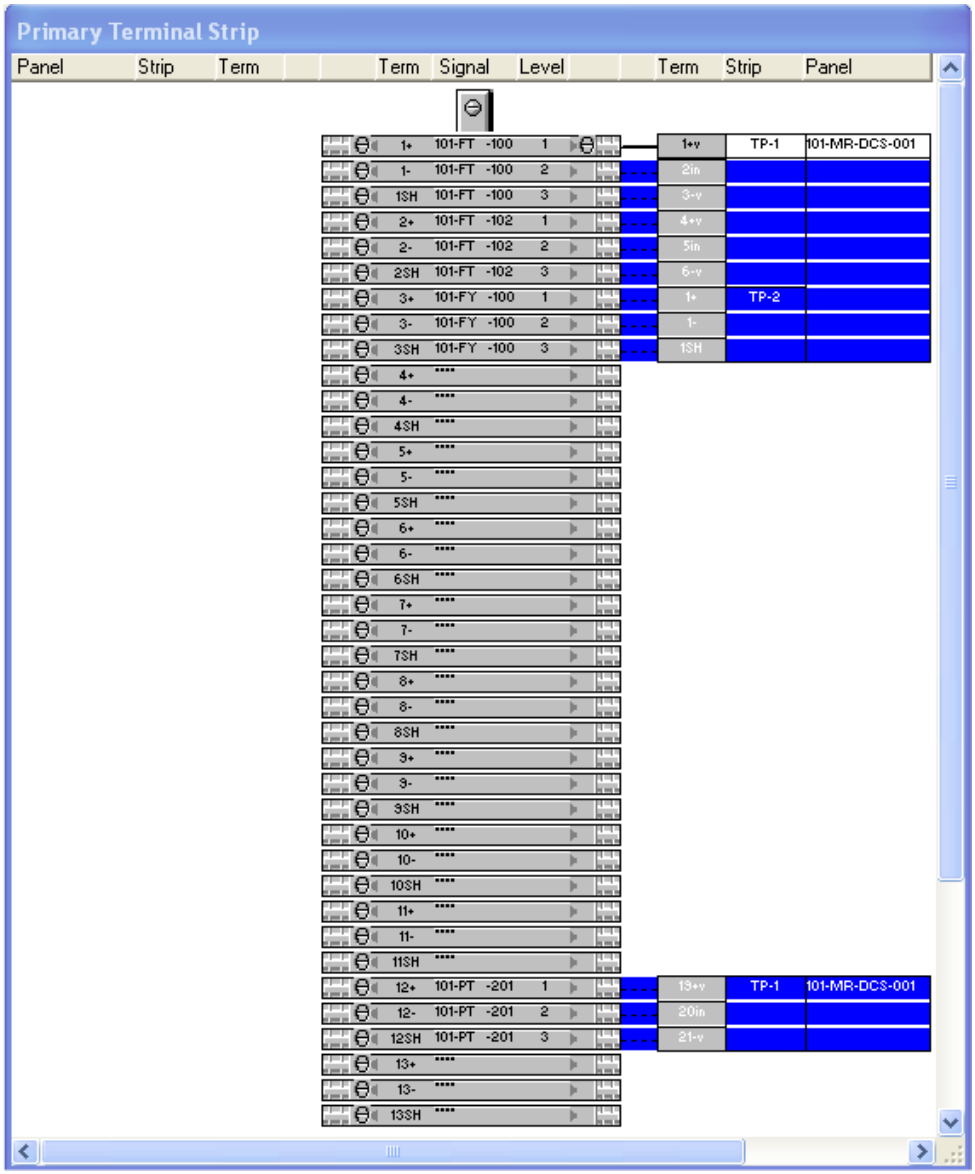
Observe how your primary and secondary terminal strips have been connected:



9. Click  to close the **Cross Wiring** window.

Cross Wire the Rest of the Signals Automatically

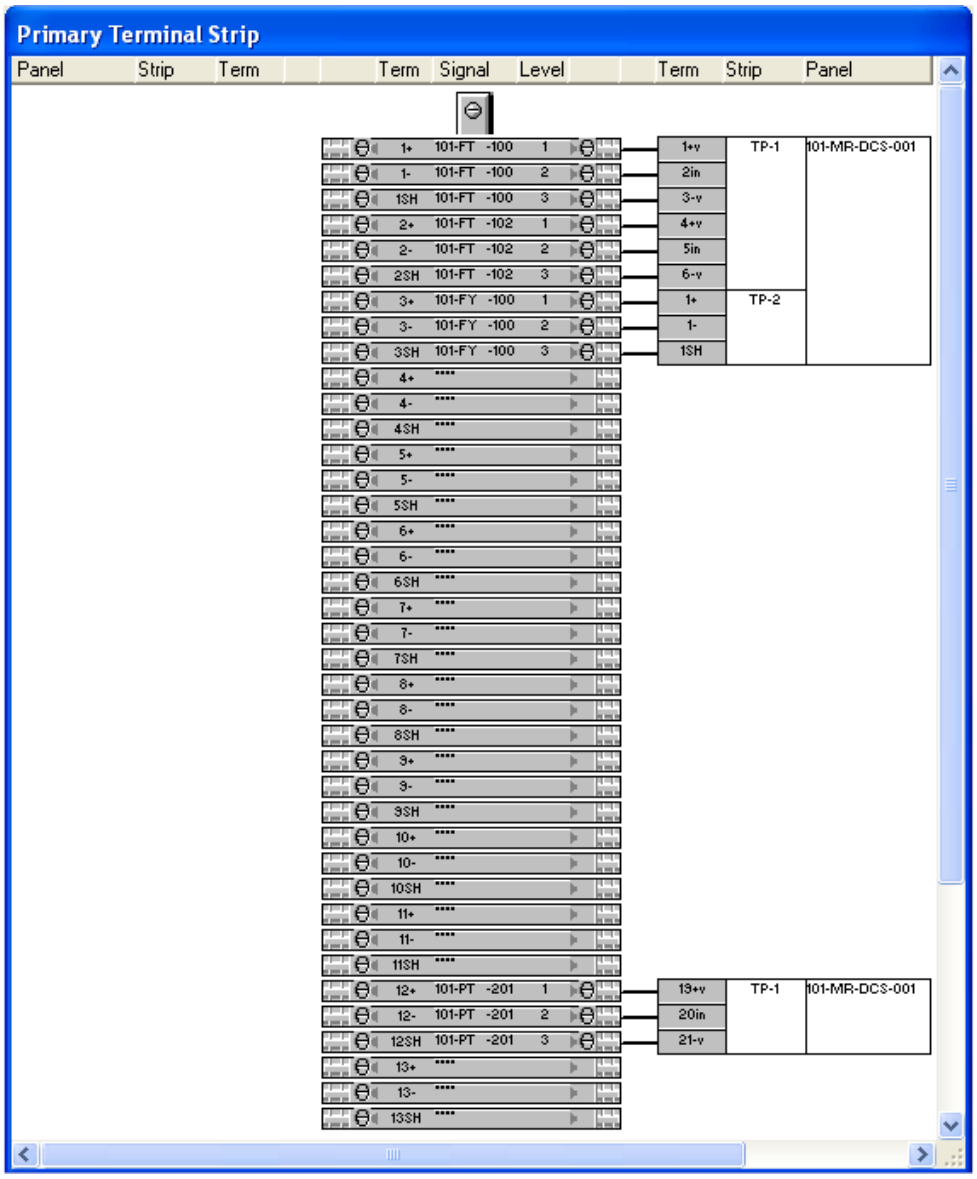
1. In the **Domain Explorer**, expand the **Panels by Category** > **Marshaling Racks** hierarchy and then select marshaling rack **101-MR-DCS-001**.
2. Right-click the selected marshaling rack and then on the shortcut menu, click **Actions** > **Cross Wiring**.
3. Under **Primary terminal strip,** make sure that terminal strip **101-MR-DCS-001**, **FT-1** is selected.
4. Under **Primary auto cross-wiring side**, select **Right**.
5. On the Cross Wiring toolbar, click the **Auto** icon to find the terminals that can be cross-wired and to show the potential cross-wiring connection by a dotted line.



TIP The software has found matching terminals **2in**, **3-v**, **4+v**, **5in**, and **6-v** in terminal strip **TP-1** belonging to marshaling rack **101-MR-DCS-001** and terminals **1+**, **1-**, and **1SH** in terminal strip **TP-2** belonging to the same marshaling rack.

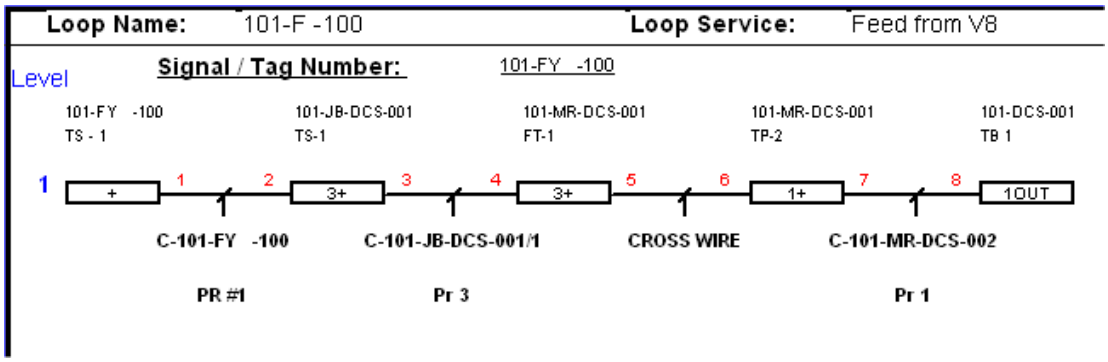
6. Click to cross wire the selected terminals.

 All the terminals have been cross-wired and the **Primary Terminal Strip** pane should appear as shown:



7. Click to close the **Cross Wiring** window.
8. In the **Domain Explorer**, right-click marshaling rack 101-MR-DCS-001 and then on the shortcut menu, click **Actions** > **Connection**.
9. In the **Connection** window, select one of the wires connected to terminal **3+** and click to open the Point-to-Point Wiring Diagram.
10. When prompted to preview the report, click **Yes**.

The diagram is displayed as shown:



As you can see, the wiring discontinuity has disappeared and the wiring continues uninterrupted from the field device to the DCS card.

11. Print out the Point-to-Point Wiring Diagram if needed and then click  to close it.
12. Click  to close the **Connection** window.

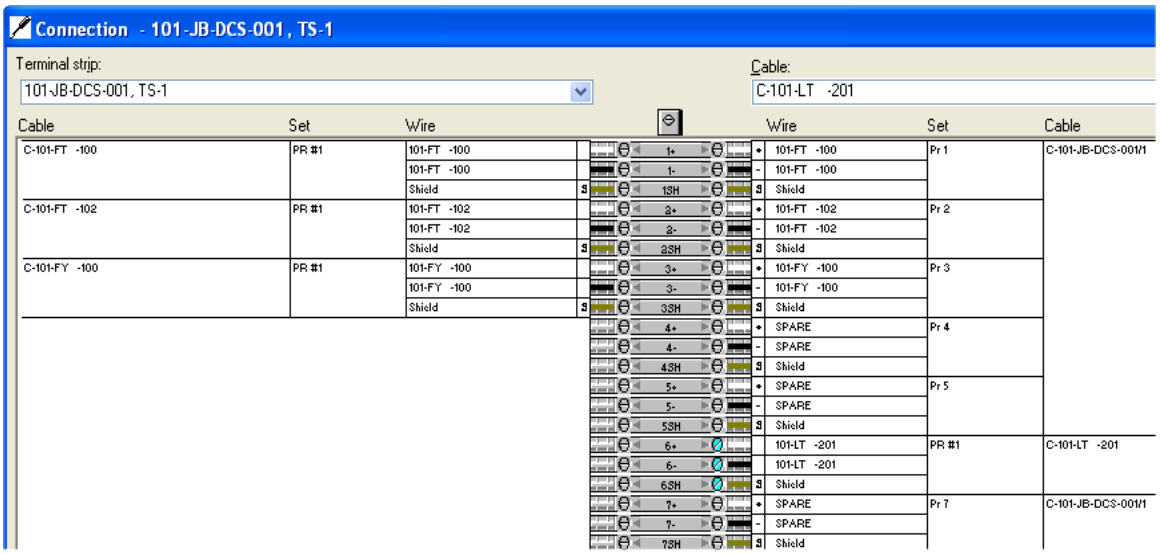
Task 11 — Adding a New Instrument to the Existing Wiring

This task shows you how you can easily add a new instrument to your existing wiring.

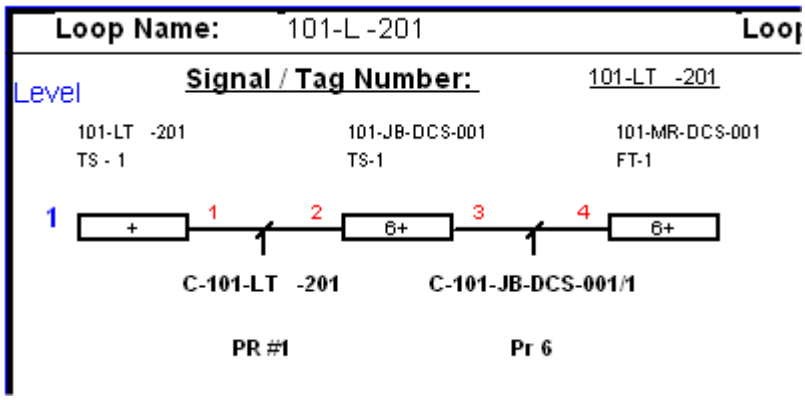
Add a New Instrument to the Existing Wiring

1. In the **Domain Explorer**, expand the **Panels by Category** > **Junction Boxes** hierarchy and then select junction box **101-JB-DCS-001**.
2. Right-click the selected junction box and then on the shortcut menu, click **Actions** > **Connection**.
3. Make sure that terminal strip **101-JB-DCS-001, TS-1** is selected in the **Terminal strip** list (there is only one terminal strip on the list).
4. In the Cables folder of Domain Explorer, select cable **C-101-LT-201** and drag it to the right side of terminal 6.
5. On the **Cable Connection Definition** dialog box, make your selections as follows:
 a. Under **Cable end to connec**t, select **End 2**.
 b. Select the connection type as **3 in a row** and select the cable set in the right data window.

6. Click **Connect** to make the connection.

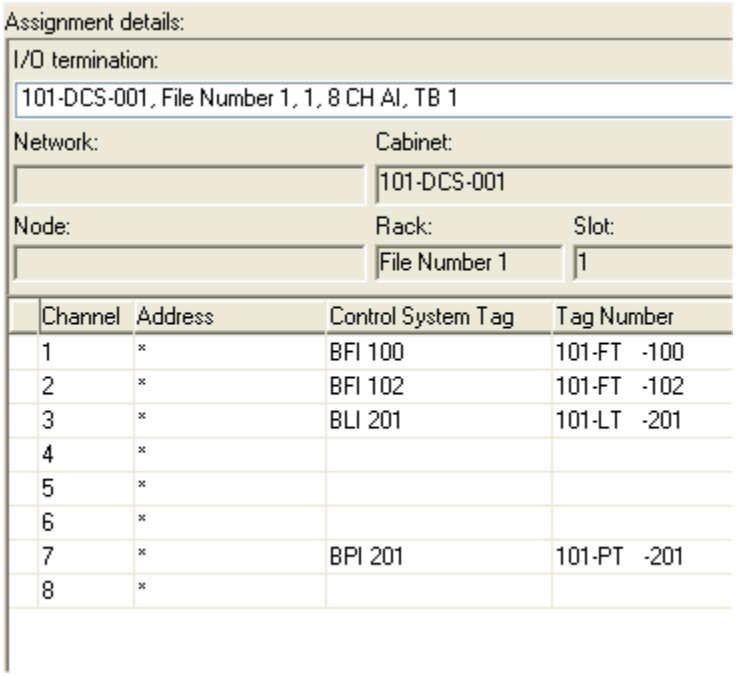


7. After the cable has been connected, open the Point-to-Point Wiring Diagram for the wire connected to terminal **6+** and view the connection. It should go straight to terminal strip **FT-1** of the marshaling rack.



8. Print out the Point-to-Point Wiring Diagram if needed and then click to close it.
9. Click to close the **Connection** window.
10. Now select **101-DCS-001** from the **DCS Panels** folder in the **Domain Explorer**.
11. Right-click the selected panel, then on the shortcut menu, click **Actions** > **I/O Assignment**.
12. On the module toolbar, click to open the **I/O Assignment Filter** dialog box.
13. Select the first three filtering criteria and click **OK**.
14. In the **I/O Assignment** window, select tag **101-LT -201** and drag it to channel 3 in the right side pane.

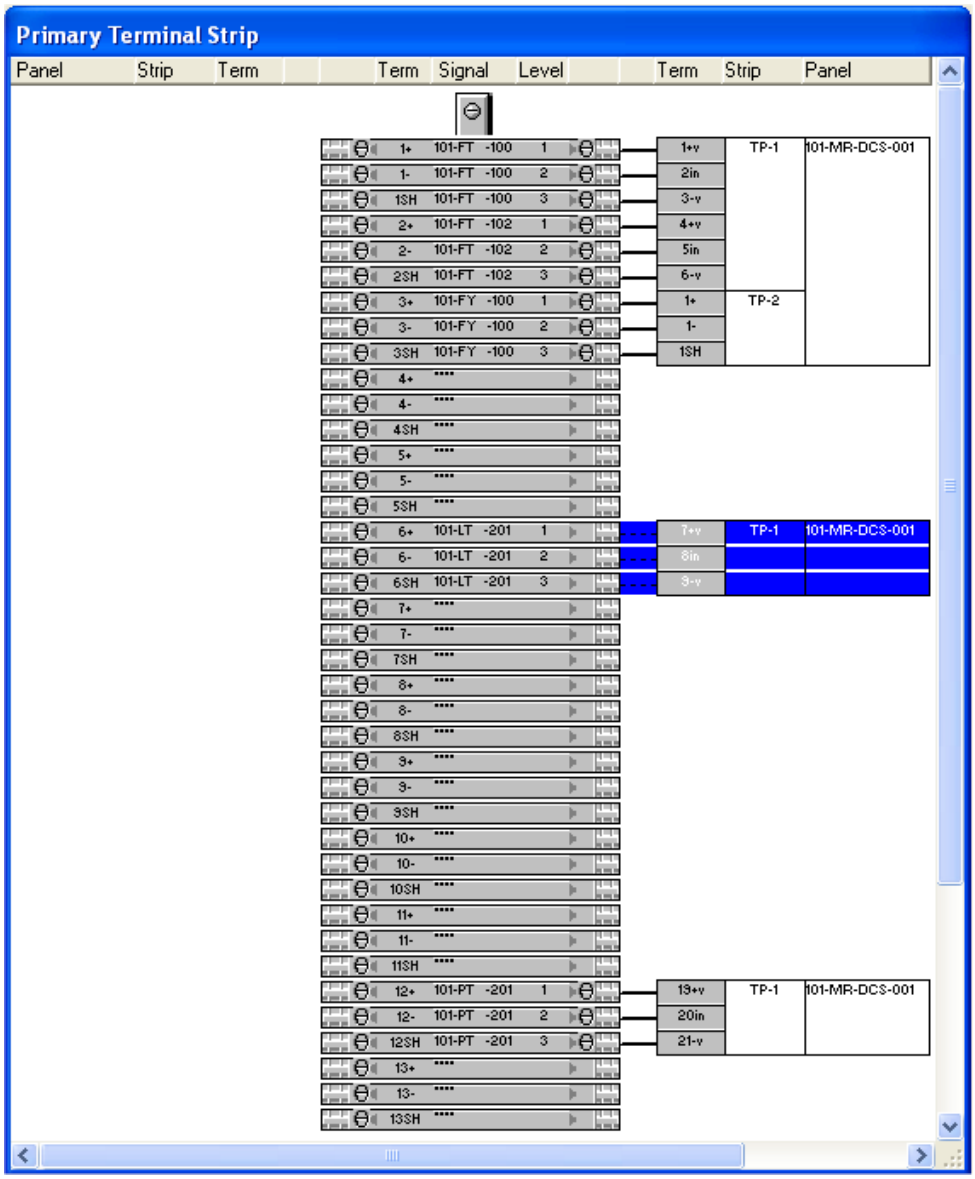
15. On the **Control System Tag Properties** dialog box, type **BLI 201** and click **OK**.



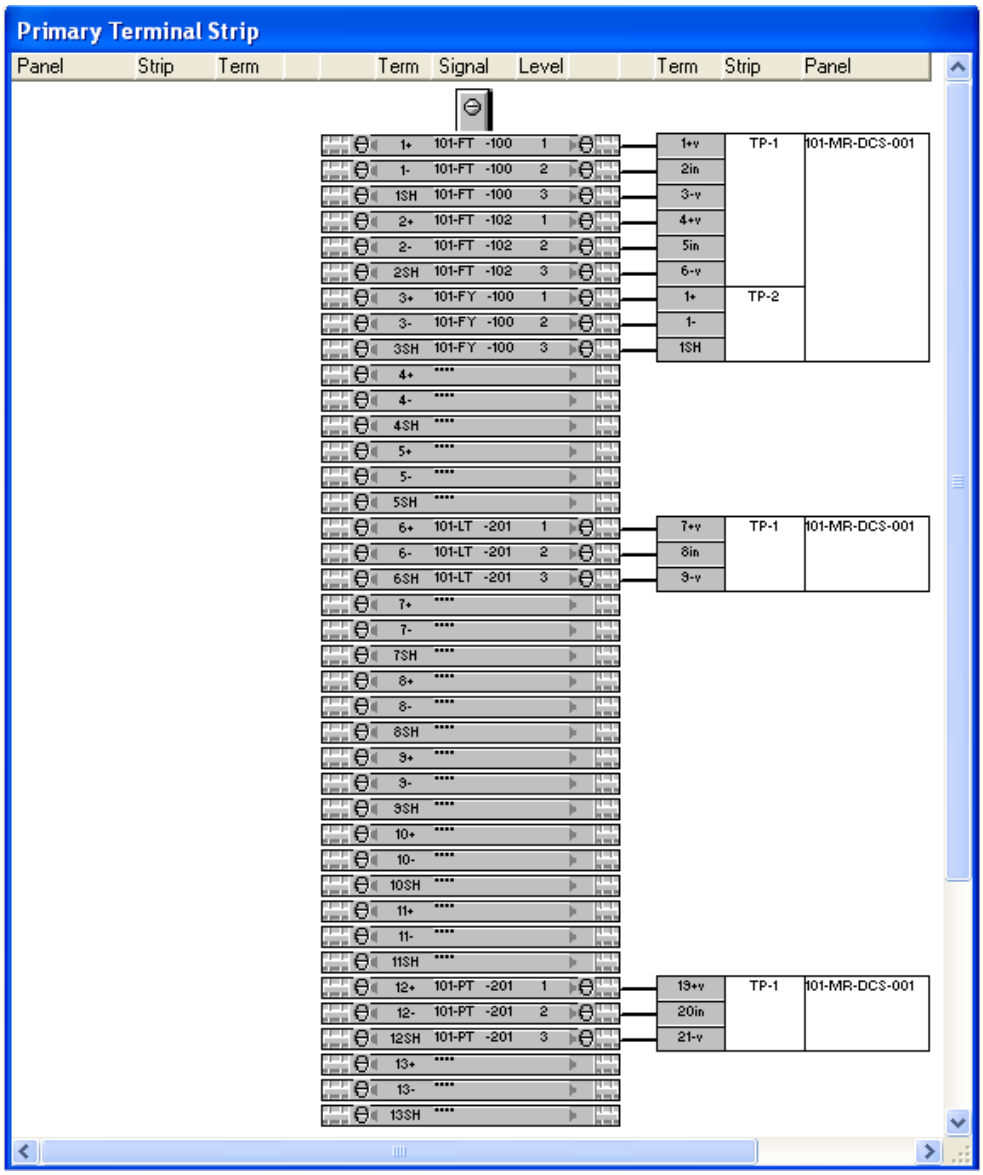
Channel	Address	Control System Tag	Tag Number
1	*	BFI 100	101-FT -100
2	*	BFI 102	101-FT -102
3	*	BLI 201	101-LT -201
4	*		
5	*		
6	*		
7	*	BPI 201	101-PT -201
8	*		

16. Now close the **I/O Assignment** window and select marshaling rack **101-MR-DCS-001**.
17. Right-click the selected panel, then on the shortcut menu, click **Actions** > **Connection**.
18. In the **Terminal strip** list, select **101-MR-DCS-001, TP-1**.
 Note that your tag wiring has been propagated from the DCS to the marshaling rack terminals.
19. Close the **Connection** window.
20. Right-click the same marshaling rack again, then on the shortcut menu, click **Actions** > **Cross Wiring**.
21. Click  to find matching terminals for automatic cross wiring.

The software finds matching terminals for automatic cross-wiring. You can view them by scrolling down in the **Primary Terminal Strip** pane:



22. Click  to perform the automatic cross wiring.



23. Close the **Cross Wiring** window.

You have successfully added a new tag to your wiring.

Task 12 — Viewing and Printing Wiring Reports

The software allows you to generate an array of wiring reports. These include various cable, panel, and I/O reports.

View and Print Various Wiring Reports

You will now generate the standard system wiring reports to understand their functions.

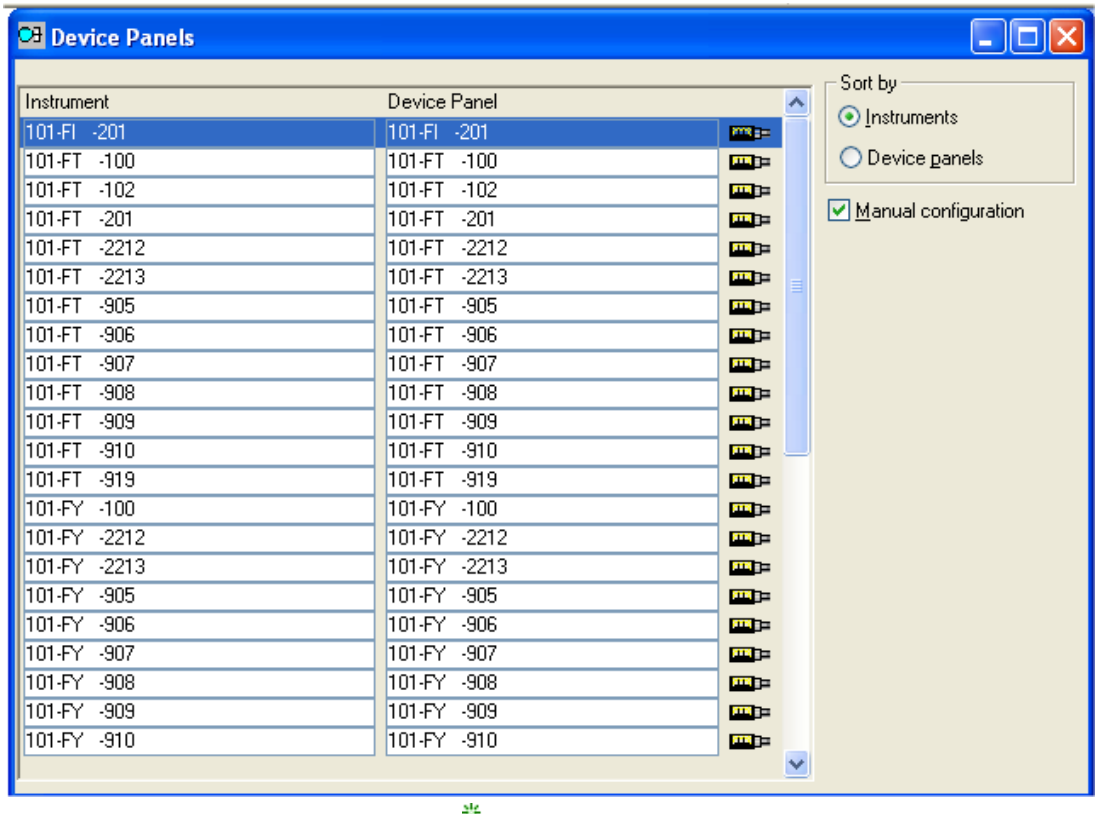
1. Open the Wiring module open and on the **Reports** menu, select the required report.
2. Where a dialog box for the report opens, enter the required criteria and click **OK**.
3. When prompted to preview the report, click **Yes**.

Task 13 — Creating Wiring Items for a New Tag

This task is appropriate if you have forgotten to set a device panel, a cable, and a connection type in the instrument profile.

Create a New Instrument Tag without an Associated Profile

1. Click on the main toolbar to open the Wiring module.
2. On the module toolbar, click to open the **Device Panels** window.

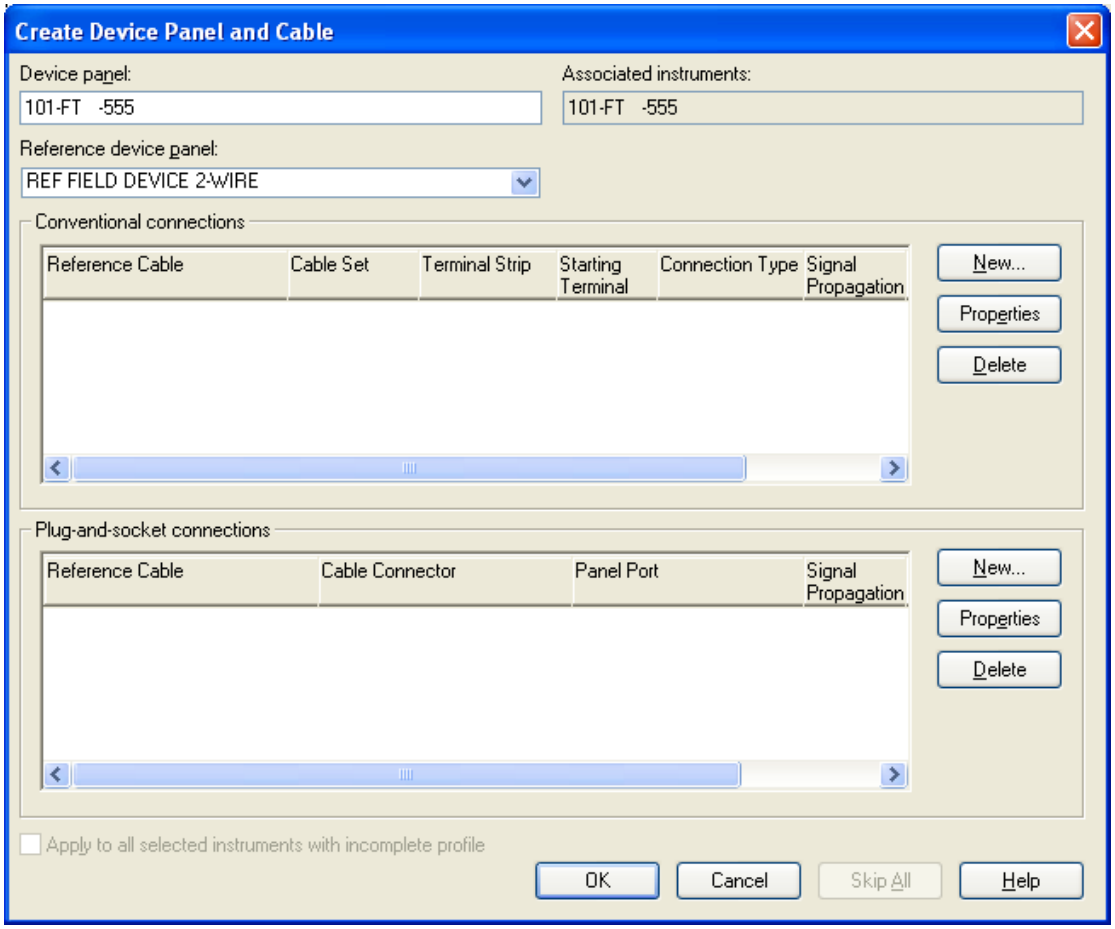


3. On the module toolbar, click **New Tag** .
4. On the **New Tag Number** dialog box, under **Tag class**, select **Conventional**.

5. Under **Tag number**, type **101-FT-555** as the new tag name and click **OK**.

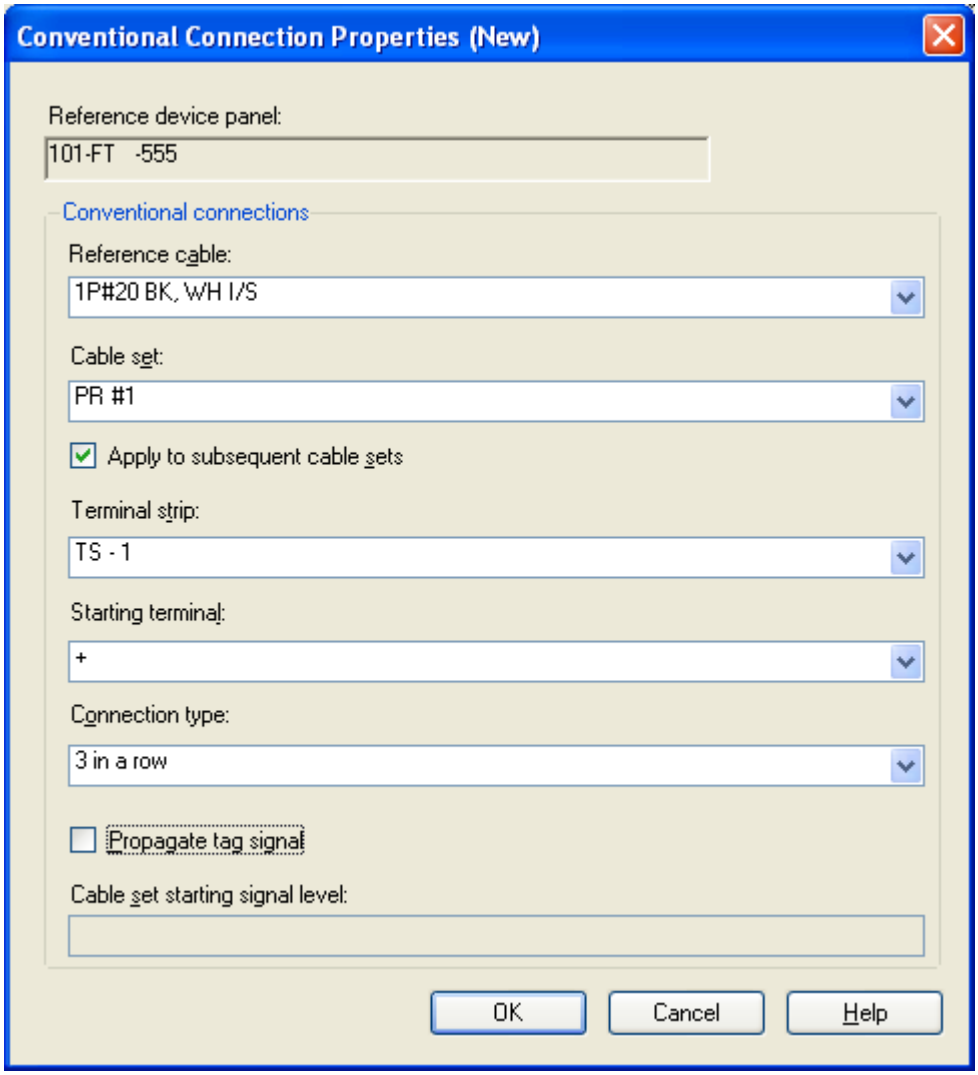
 TIP If the **Select Instrument Type** dialog box opens, select **FT FLOW TRANSMITTER** and click **OK**.

6. On the **Loop Name** dialog box, accept **101-F-555** and click **OK**.
7. On the **Loop Number Properties** dialog box, accept all the definitions and click **OK**.
8. On the **Create Device Panel and Cable** dialog box, from the **Reference device panel** list, select **REF FIELD DEVICE 2-WIRE**.

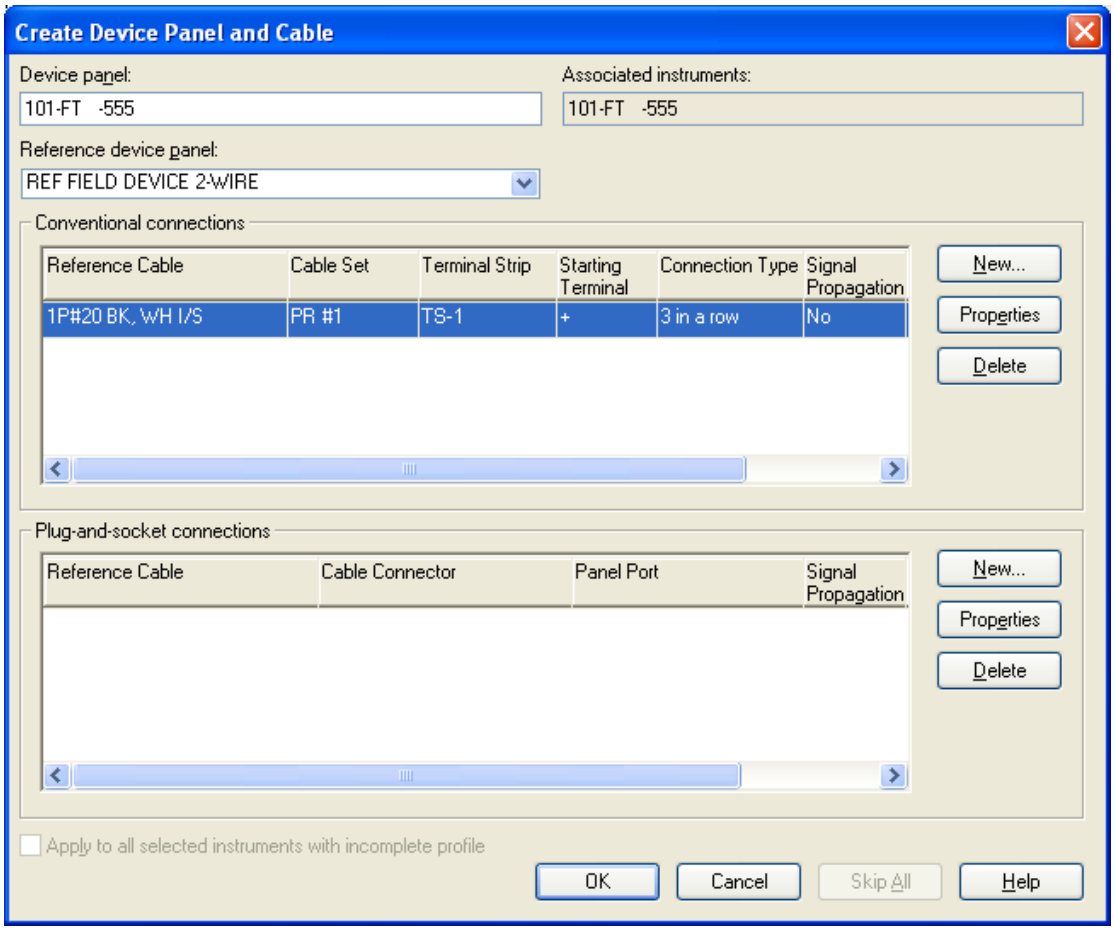


9. In the **Conventional connections** group box, click **New**.

10. On the **Conventional Connection Properties** dialog box, enter data as shown below, and then click **OK**.



The **Create Device Panel and Cable** dialog box should now appear as shown:

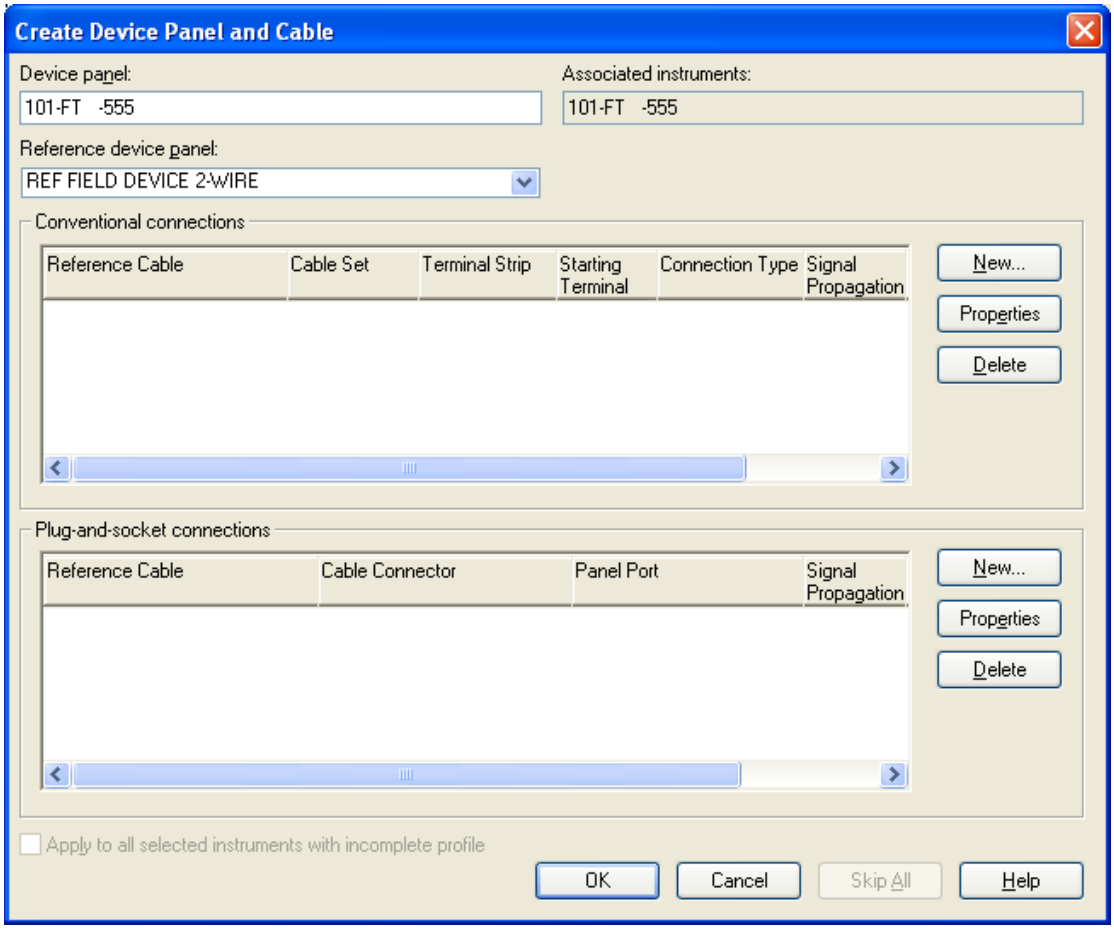


11. Click **OK** to close the **Create Device Panel and Cable** dialog box.
12. On the **Tag Number Properties** dialog box, accept the given values and click **OK**.
13. Click  to close the **Device Panels** window.
14. In the **Domain Explorer**, click , and find the new device panel and cable you just created (in their respective folders).

Add a Device Panel and a Cable to the Tag

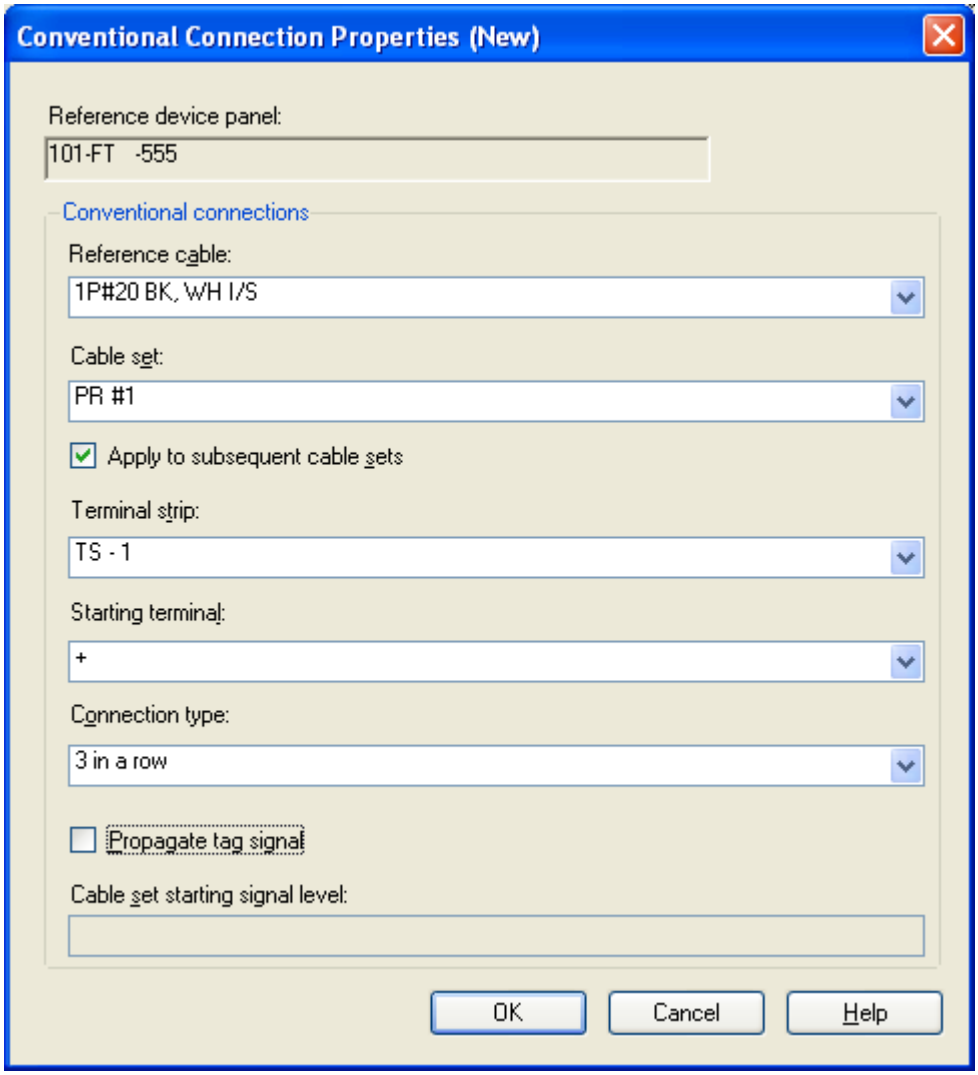
1. In the Domain Explorer, select the instrument tag that you created in the last activity.
2. Right-click the tag and on the shortcut menu, click **Actions** > **Create Device Panel and Cable**.

3. On the **Create Device Panel and Cable** dialog box, from the **Reference device panel** list, select **REF FIELD DEVICE 2-WIRE**.

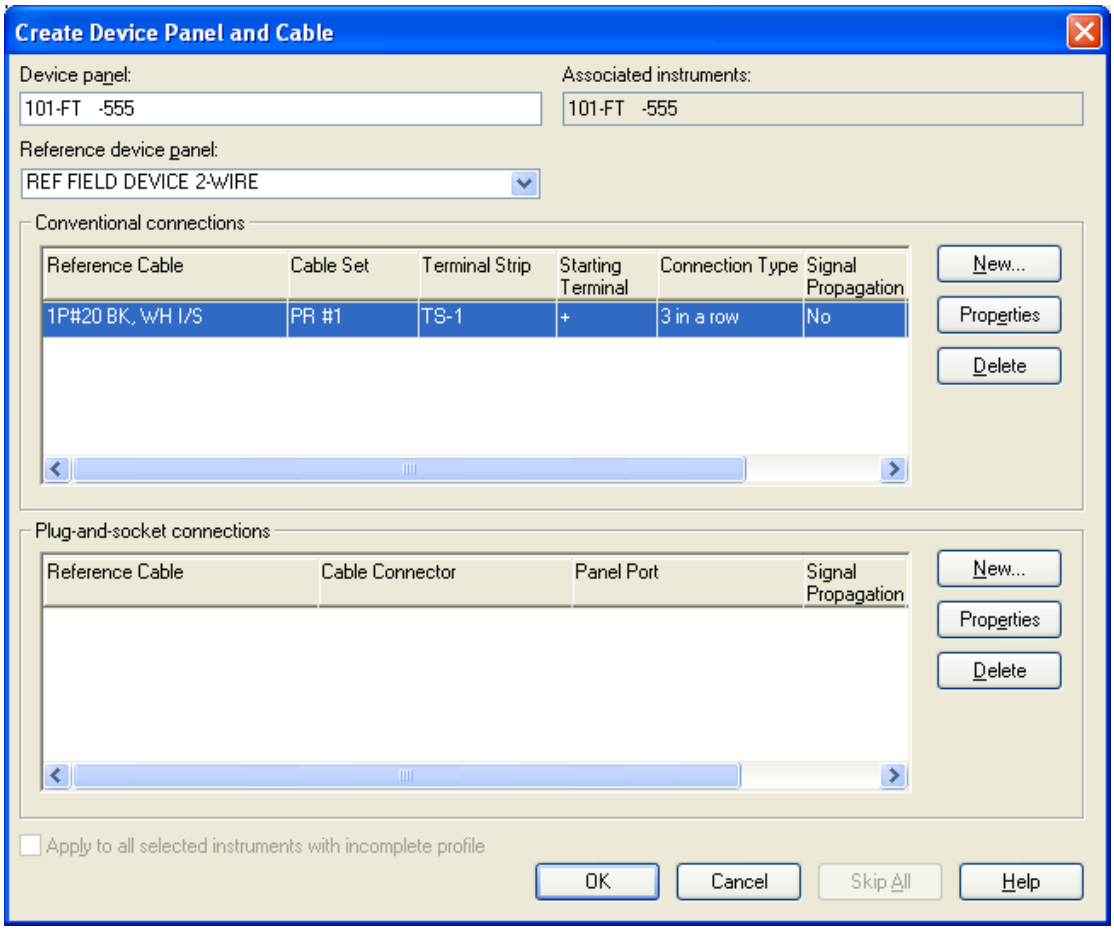


4. In the **Conventional connections** group box, click **New**.

5. On the **Conventional Connection Properties** dialog box, enter data as shown below, and then click **OK**.



The **Create Device Panel and Cable** dialog box should now appear as shown:



6. Click **OK** to close the **Create Device Panel and Cable** dialog box.
7. On the **Tag Number Properties** dialog box, accept the given values and click **OK**.
8. Click  to close the **Device Panels** window.
9. In the **Domain Explorer**, click , and find the new device panel and cable you just created.

Task 14 — Wiring Design that Includes a Barrier

You will now learn how to create a barrier and include it in your wiring design. You will do so by adding a barrier terminal strip in marshaling rack **101-MR-DCS-001**. To create a barrier you will use the **Apparatus** feature in the Wiring module. **Apparatus** pertains to I/O devices of different kinds. These devices can be used for various purposes, for example, devices that accept wires and signals, barriers, opto-couplers, relays, repeaters, and isolators.

Adding a new apparatus to your wiring starts with the configuration of a new apparatus group:

- Setting the number of apparatuses required.
- Defining the apparatus terminal numbering, pattern, signal propagation, and color of the terminals.

You create a new apparatus (in the exact way you create a terminal strip) after completing the new apparatus group configuration. You can also create a new apparatus based on any existing configuration. You can modify or delete an apparatus configuration. Once a new apparatus has

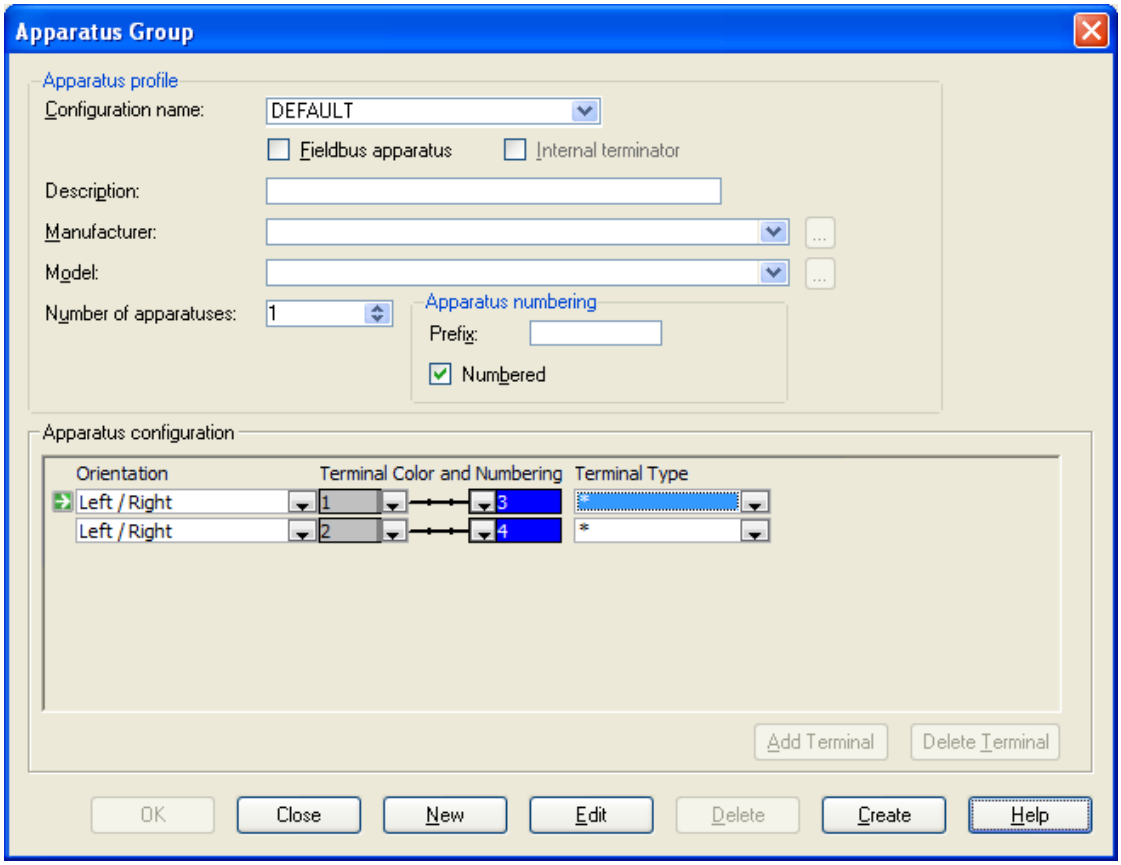
been added, it is displayed in the **Domain Explorer** showing the terminals belonging to the apparatus.

NOTE You can have regular terminals and apparatus terminals on the same strip.

Define a New Apparatus Configuration

You need to create an apparatus configuration before you can add an apparatus to a selected strip. When configuring an apparatus, you define the apparatus profile (name, description, manufacturer, and model), the number of apparatuses, and the apparatus terminal configuration.

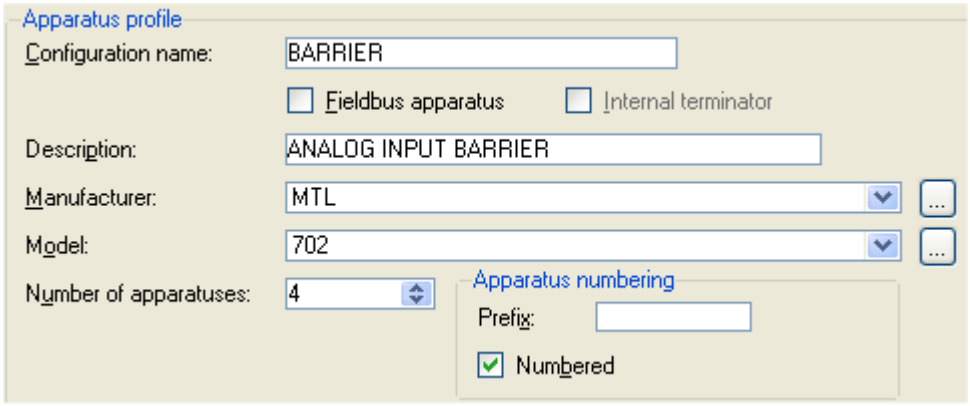
1. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **Marshaling Racks** folder.
2. Right-click marshaling rack **101-MR-DCS-001** and on the shortcut menu click **New** > **Apparatus Group**.



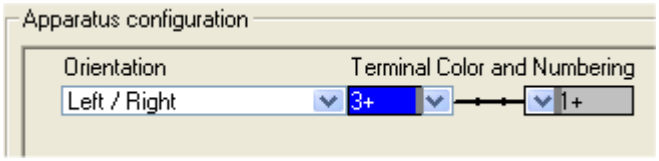
3. On the **Apparatus Group** dialog box, click **New** to create a new apparatus configuration.
4. In the **Configuration name** data field, type **BARRIER**.
5. Enter the description **ANALOG INPUT BARRIER** in the **Description** data field.
6. From the **Manufacturer** list, select **MTL**. If this value is not available from the list, click ... to add it to the list, and then select it from the list.
7. From the **Model** list, select **702**. If this value is not available from the list, click ... to add it to the list, and then select it from the list.

8. In the **Number of apparatuses** spin box, select **4** to have a block of four apparatuses to be added by default when adding a new apparatus to a panel.
9. In the **Apparatus numbering** group box, select the **Numbered** check box.

 The **Apparatus profile** group box should now appear as shown:

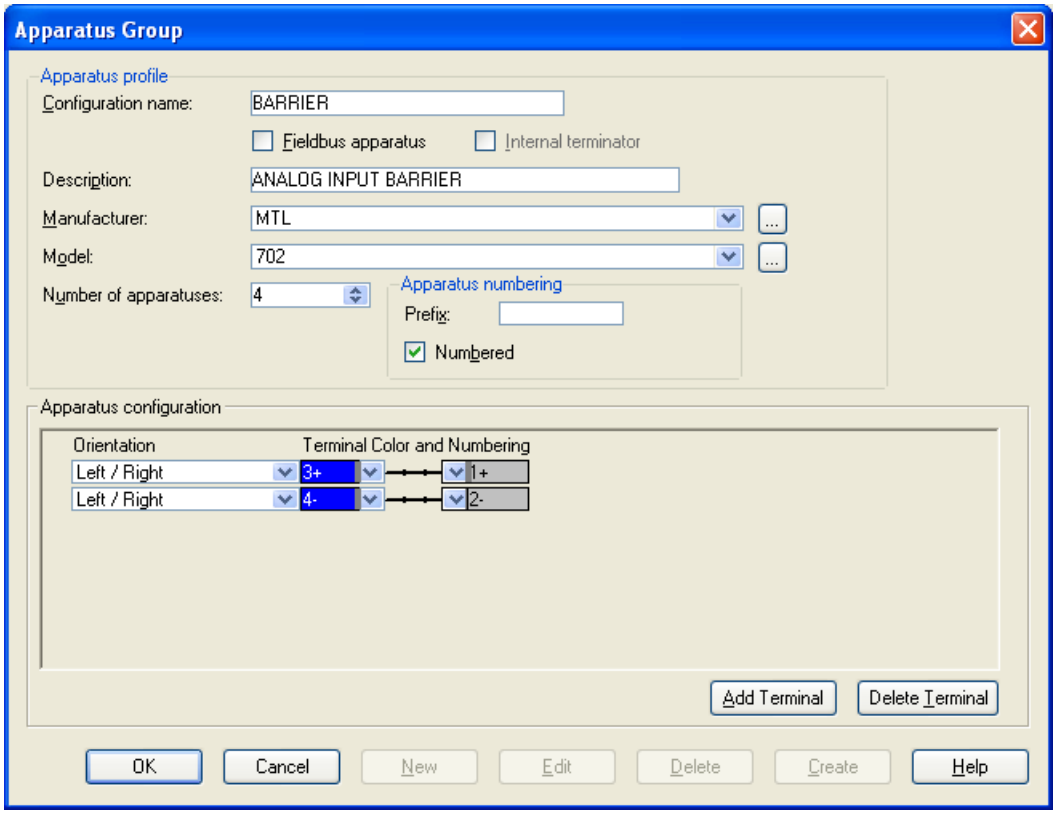


10. To define the apparatus terminal configuration, do the following in the **Apparatus configuration** group box:
 a. From the **Orientation** list, select **Left / Right** to represent an apparatus that has two sides — an input and an output.
 b. Under **Terminal Color and Numbering**, click in the left text box and type **3+** as the terminal name.
 c. From the left **Terminal Color and Numbering** list, select **Blue** as the color of the left terminal side.
 d. Click in the right **Terminal Color and Numbering** text box and type **1+** as the terminal name.
 e. From the right **Terminal Color and Numbering** list, select **Gray** as the color of the right terminal side.



11. Click **Add Terminal** to add another terminal to this apparatus configuration.
12. Define the second terminal as follows:
 a. From the **Orientation** list, select **Left / Right**.
 b. Click in the left **Terminal Color and Numbering** text box and type **4**– as the terminal name.
 c. From the left **Terminal Color and Numbering** list, select Blue as the color of the left terminal side.
 d. Click in the right **Terminal Color and Numbering** text box and type **2** – as the terminal name.
 e. From the right **Terminal Color and Numbering** list, select **Gray** as the color of the right terminal side.

13. Ensure that your dialog box appears as follows before you proceed with the apparatus creation.



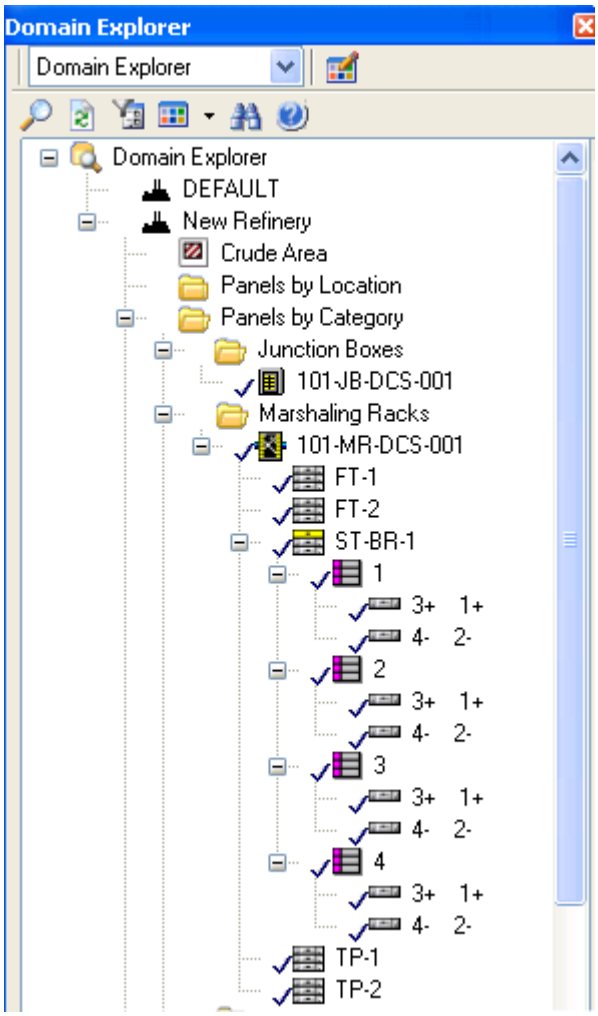
14. Click **OK** to save the new apparatus configuration.

Create a New Apparatus and Make All the Connections

Having defined a new apparatus configuration, you will now add a new apparatus to marshaling rack 101-MR-DCS-001. When adding a new apparatus, you will also create a new strip where this apparatus will be located. It is also possible to add an apparatus to an existing strip if required.

1. On the **Apparatus Group** dialog box, click **Create**.
2. On the **Terminal Strip Properties** dialog box, under **Terminal strip**, type **ST-BR-1** and click **OK**.

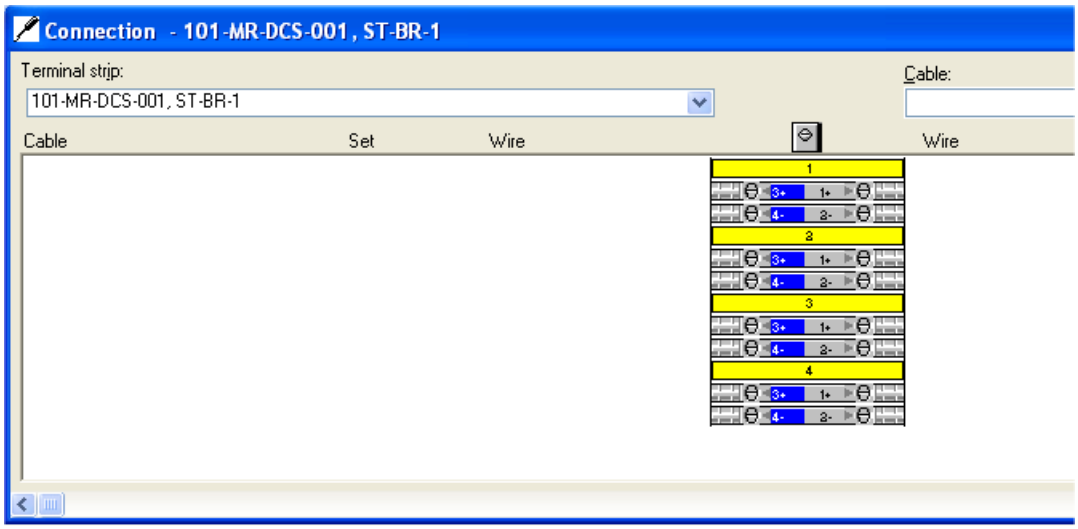
You have successfully created a new terminal strip with four barriers in marshaling rack **101-MR-DCS-001**. Expanding this marshaling rack in the **Domain Explorer** will show you the new terminal strip and its four barriers:



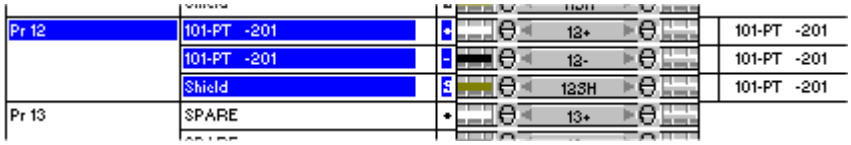
Next you will view the connection details of marshaling rack 101-MR-DCS-001.

3. In the **Domain Explorer**, right-click terminal strip **ST-BR-1**, then on the shortcut menu, click **Actions** > **Connection**.

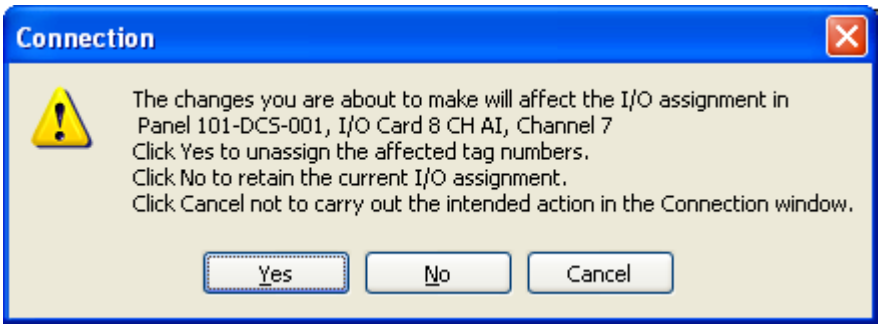
The **Connection** window opens displaying the information as shown:



4. Close the **Connection** window and then in the **Domain Explorer**, under marshaling rack **101-MR-DCS-001**, right-click terminal strip **FT-1** and then click **Actions** > **Connection**.
5. In the **Connection** window, do the following to disconnect **Pr 12** on the left side of the marshaling rack:
 a. Click cable set **Pr 12** to select the cable set and its wires:



 b. Click  on the **Connection** window toolbar to disconnect the selected wires.
 c. Click **No** in the following message to retain the existing I/O assignment.

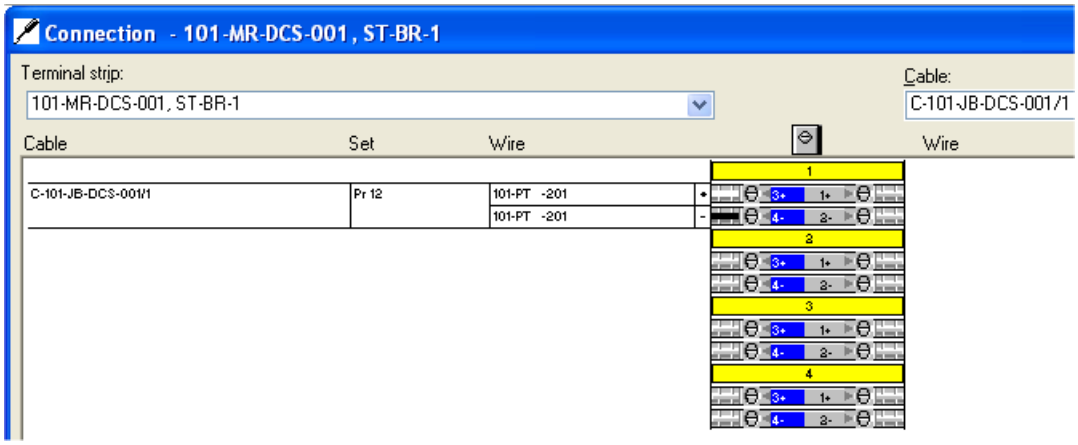


6. Close the **Connection** window.
7. In the **Domain Explorer**, expand marshaling rack **101-MR-DCS-001** and right-click terminal strip **ST-BR-1**. On the shortcut menu, click **Actions** > **Connection**.
8. Reconnect **Pr 12** on the left (marshaling) side of the terminal strip as follows:
 a. In the **Domain Explorer**, expand the **Cables** folder, scroll down to cable **C-101-JB-DCS-001/1** and click it to display its cable sets.
 b. Click cable set **Pr 12** to display its wires.
 c. Drag the upper **101-PT-201** wire to the first (**3+**) terminal.
 d. On the **Wire Connection Options** dialog box, accept the **End 2** option and click **OK**.

e. Drag the lower **101-PT-201** wire to the first (**4-**) terminal.

f. On the **Wire Connection Options** dialog box, accept the **End 2** option and click **OK**.

Now the **Connection** window for **101-MR-DCS-001** should appear as shown:



9. Close the **Connection** window and in the **Domain Explorer** expand 101-DCS-001.

 Now all that remains to complete the signal propagation is to perform cross wiring in the marshaling rack between the appropriate terminals on strips **ST-BR-1 and FT-1**.

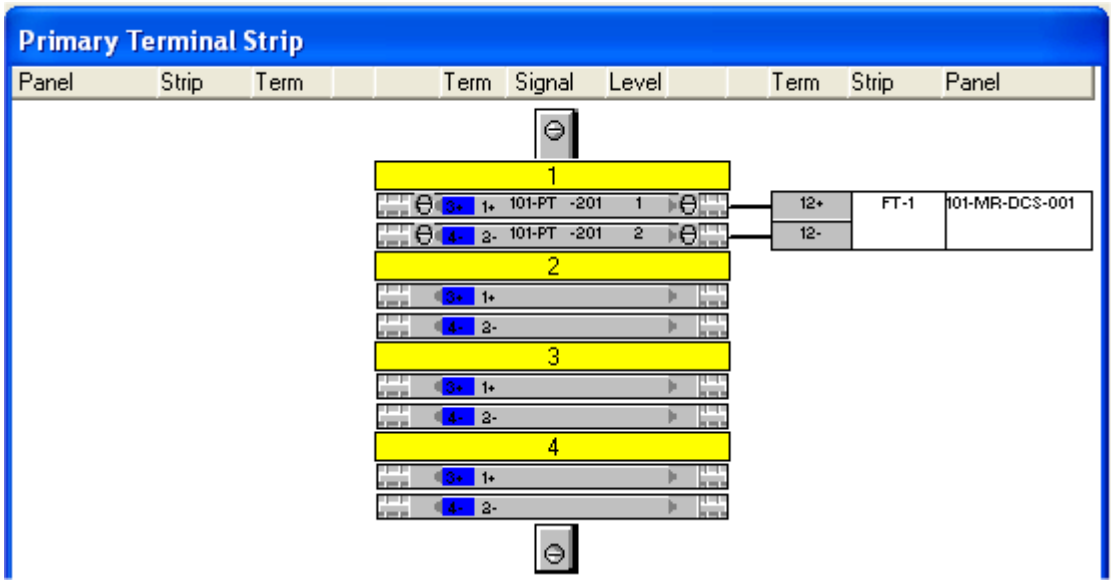
10. Right-click marshaling rack **101-MR-DCS-001** and on the shortcut menu, click **Actions** > **Cross Wiring**.

11. In the **Cross Wiring** window, select primary strip **101-MR-DCS-001**, **ST-BR-1** and secondary strip **101-MR-DCS-001, FT-1**.

12. Click  to find the terminals on terminal strip **FT-1** that can be cross-wired with terminal strip **ST-BR-1**.

13. Accept the suggested defaults and click  to complete the cross wiring.

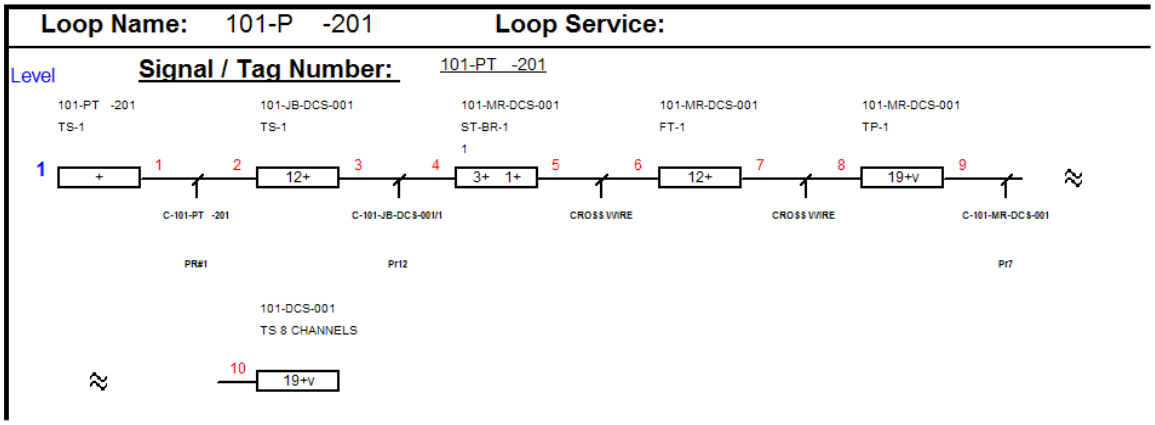
 At the end of the cross wiring procedure, the **Primary Strip** pane should appear as shown:



This completes the wiring design for loop **P-201**.

14. To view the new connections, select a terminal and on the **Cross Wiring** toolbar, click .

The Point-to-Point Wiring Diagram appears as shown:



Task 15 — Wiring Design for Other Loop Types

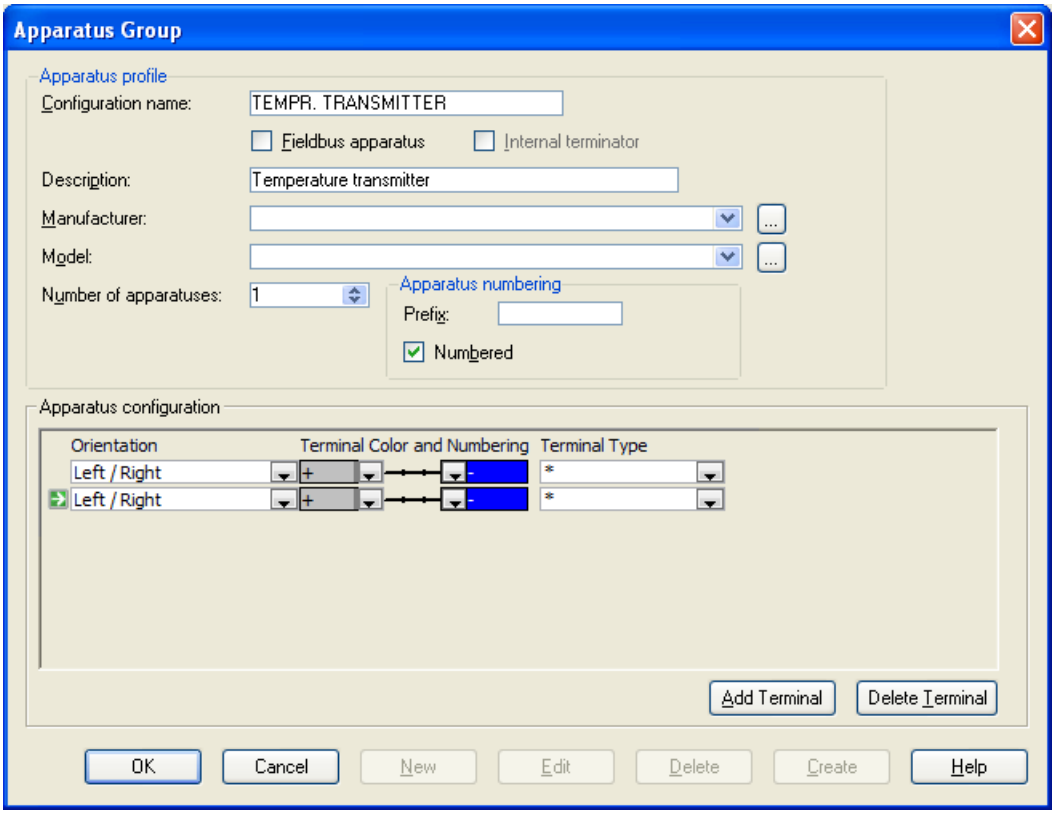
In this section, you will learn how to create wiring for loops that include local indicators, jumpers, and temperature transmitters and view the related enhanced loop drawings that have been created. We assume that you have already become familiar with the wiring procedures required for this task. Refer to the sections wherever indicated to refresh your memory.

Design the Wiring for Loop T-202

You will now create all the required wiring that constitutes loop 101-T -202.

1. Press **F8** to open the **Reference Explorer**.
2. Expand the **Panels** folder and create a new conventional device panel named **TEMPERATURE TRANSMITTER**.
3. Create an apparatus for this device panel using the following definitions:

a. On the **Apparatus Group** dialog box, define a new profile and configuration with 2 terminals as shown.

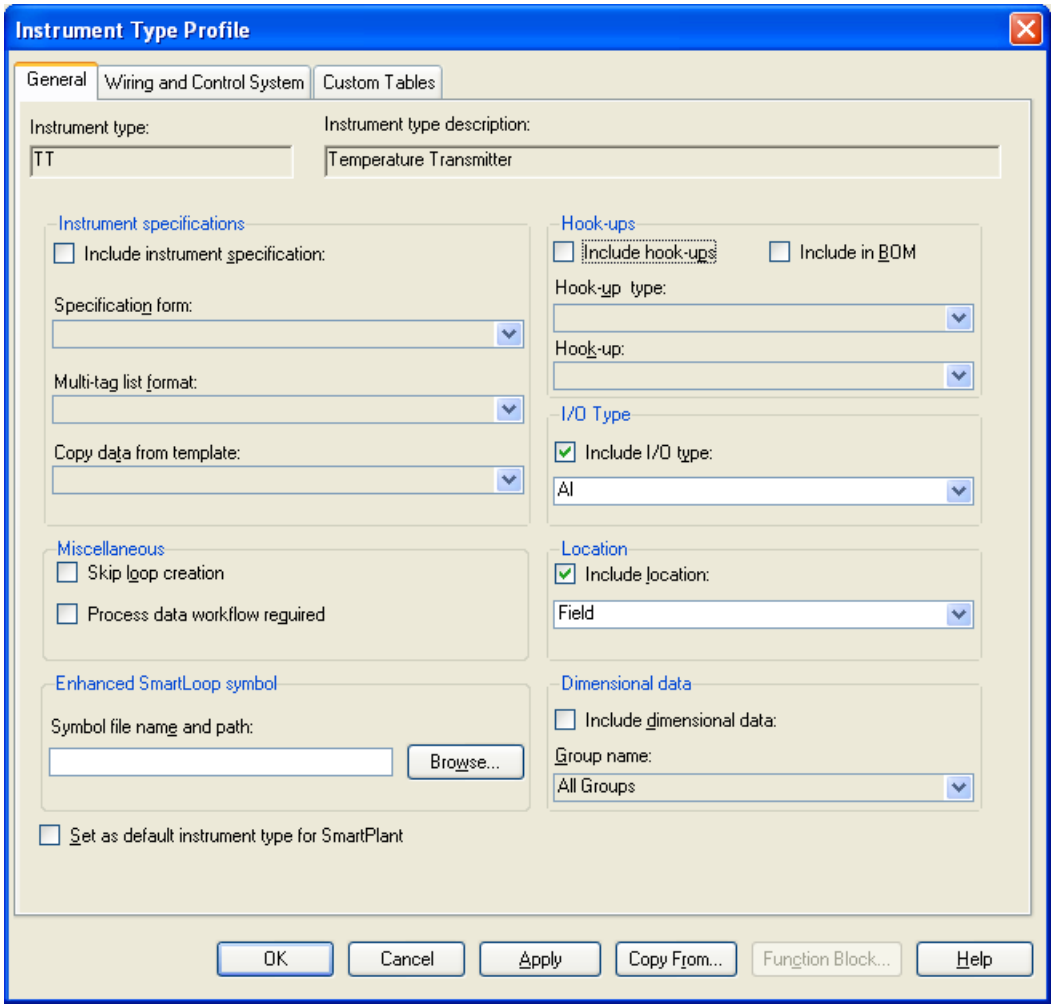


b. Click **OK** and then **Create**.

c. On the **Terminal Strip Properties** dialog box, name the terminal strip TS and click **OK**.

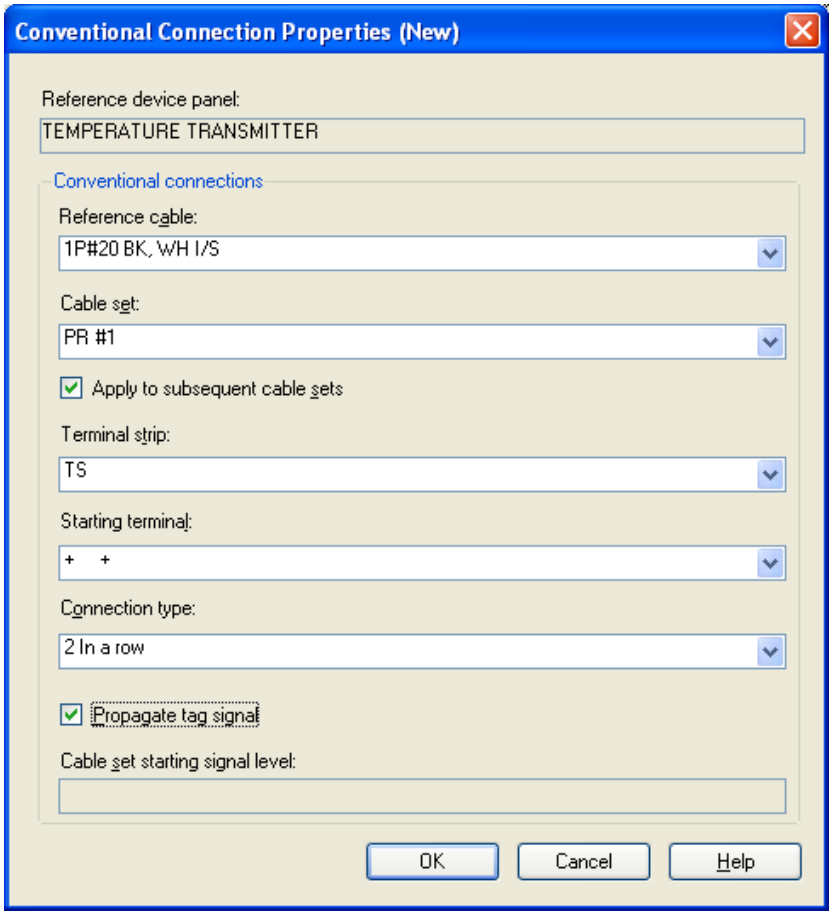
4. Open the Instrument Index module. For details of how to create instrument types, see *Define Instrument Types* (on page 43).

5. Modify the instrument type profile for **TT (Temperature Transmitter)**, making sure that the definitions on the **General** tab are the same as those shown.



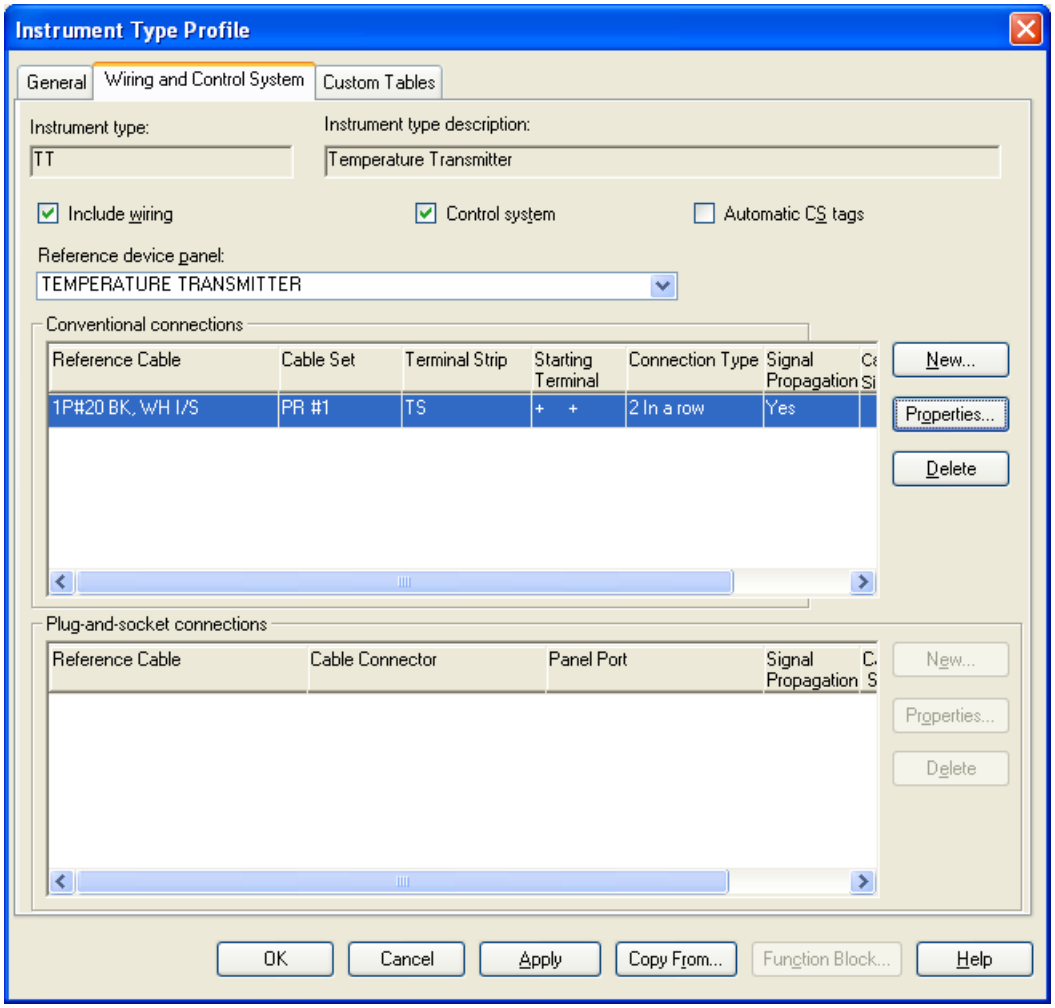
6. On the **Wiring and Control System** tab, select the **Include wiring** and **Control system** check boxes.
7. From the **Reference device panel** list, select **TEMPERATURE TRANSMITTER**.

8. In the **Conventional connections** group box, click **New**, and then complete the **Conventional Connection Properties** dialog box as shown.



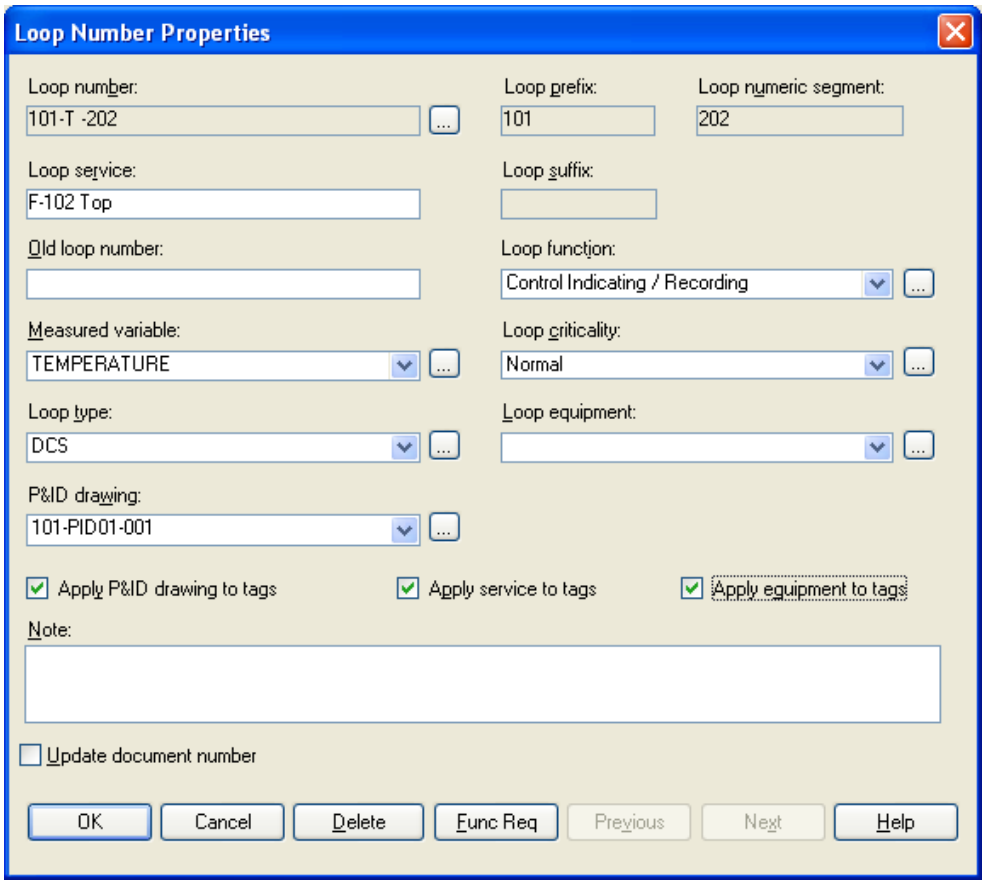
9. Click **OK** to close the **Conventional Connection Properties** dialog box.

10. Examine the definitions on the **Wiring and Control Systems** tab of the **Instrument Tag Profile** dialog box and click **OK**.



11. Click **OK** in the **Instrument Types** dialog box.
12. Press **F7** to open the **Domain Explorer**.
13. Expand the hierarchy **New Refinery** > **Crude Area** > **Crude Unit 1**.
14. Right click the **Loops** folder and then on the shortcut menu, click **New** > **Loop**.

15. Create a new loop **T-202**, using the definitions shown.

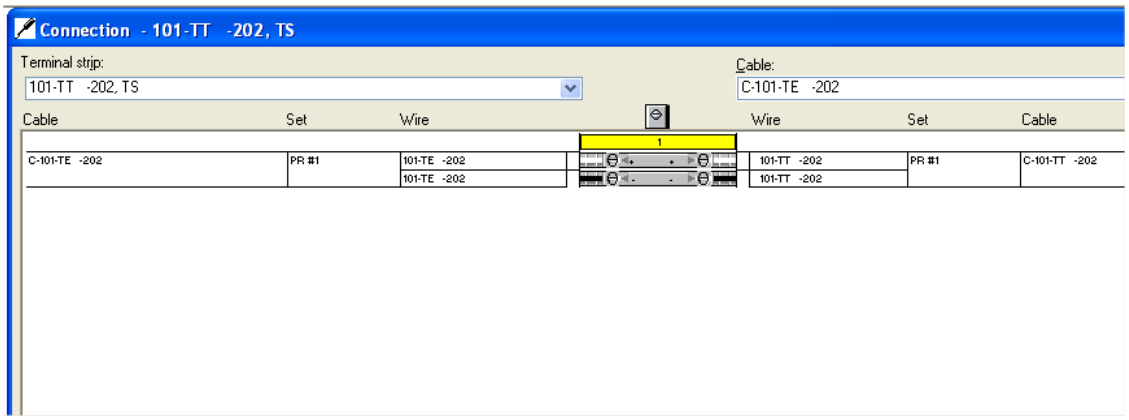


16. When prompted to create new tag numbers associated with the new loop, click **Yes** and create the following tag numbers:
 - **TE-202** - Use the **TE-THERMOCOUPLE** instrument type. After defining the new tag number, click **New** on the **Tag Number Properties** dialog box to create another tag number.
 - **TT-202** - Use the **TT- Temperature Transmitter** instrument type.
 - **TY-202** - Use the **TY-I/P Transducer** instrument type.

 You can find details of loop and tag number creation in the Instrument Index module tutorial.
17. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **Device Panels** folder.
18. Right-click device panel **101-TT-202** and then on the shortcut menu, click **Actions** > **Connection**.

 TIP If an item that you created is not visible on the tree, select the root folder (**Domain Explorer**) and click .
19. Make the device panel connections as follows:
 a. In the **Connection** window, make sure that **101-TT-202, TS** is selected on the **Terminal strip** list.
 b. In the **Domain Explorer**, expand the **Cables** folder and drag cable **C-101-TE-202** to the left side of the input terminal strip (**TT-IN**) using **2 in a row** as connection type. To learn how to make connections, see *Task 7 — Making Connections* (on page 182).

The **Connection** window should appear as shown:



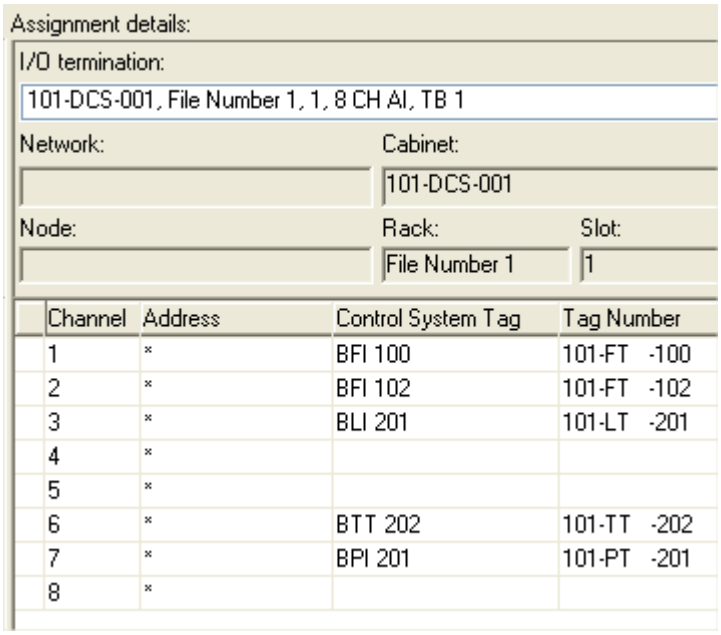
c. Close the **Connection** window.

20. Make the junction box connections as follows:
 a. Select junction box **101-JB-DCS-001** and open the **Connection** window.
 b. Connect cable **C-101-TT-202** at the left side of strip **TS-1**, starting at terminal **13+** with **3 in a row** as the connection type.

 Observe that the tag number signals have been propagated to the right side of the terminals.
21. Select the wire connected to the right of terminal **13+** and click to display the connection at the other end of the wire to marshaling rack **101-MR-DCS-001**, strip **FT-1** (This connection was already made in a previous wiring task in this tutorial).
22. Close the **Connection** window for each panel.

 You have now completed the wiring up to the marshaling rack. Next, you will define the DCS connections and do the cross wiring in the marshaling rack.
23. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **DCS Panels** folder.
24. Right-click **101-DCS-001** and then on the shortcut menu, click **Actions** > **I/O Assignment**.
25. In the **I/O Assignment** window, click on the module toolbar.
26. On the **I/O Assignment Filter** dialog box, select the first three check boxes in the **Filter for** group box and click **OK**.

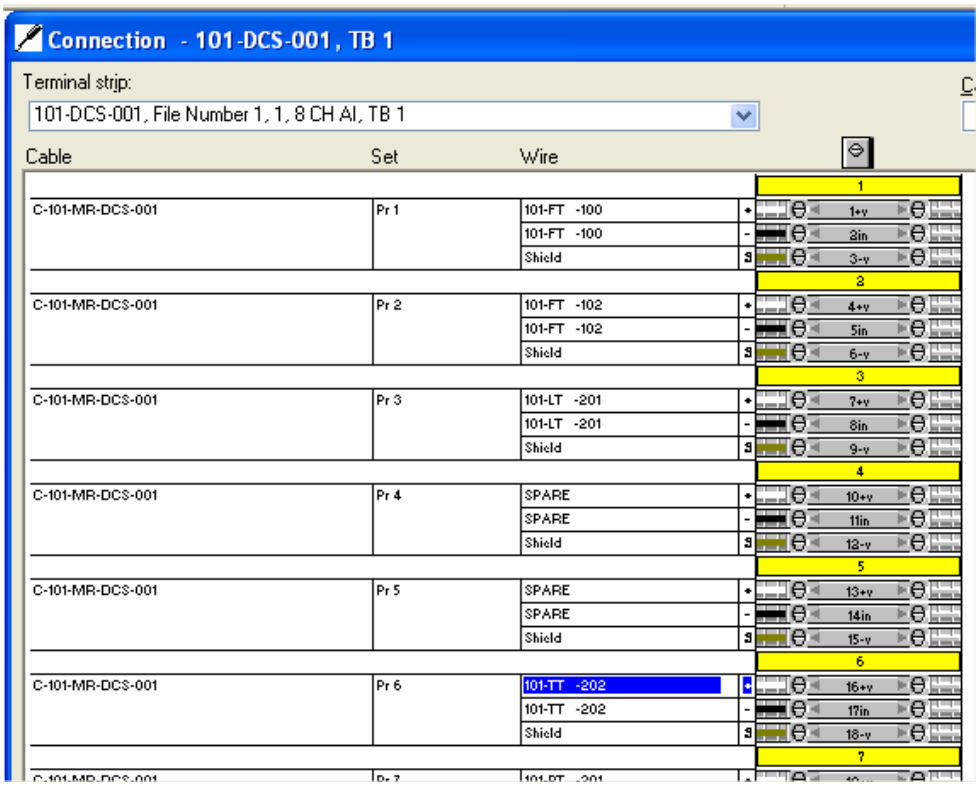
27. In the **I/O Assignment** window, assign **TT-202** to terminal strip **TB 1**, channel **6** and rename the control system tag **BTT 202**.



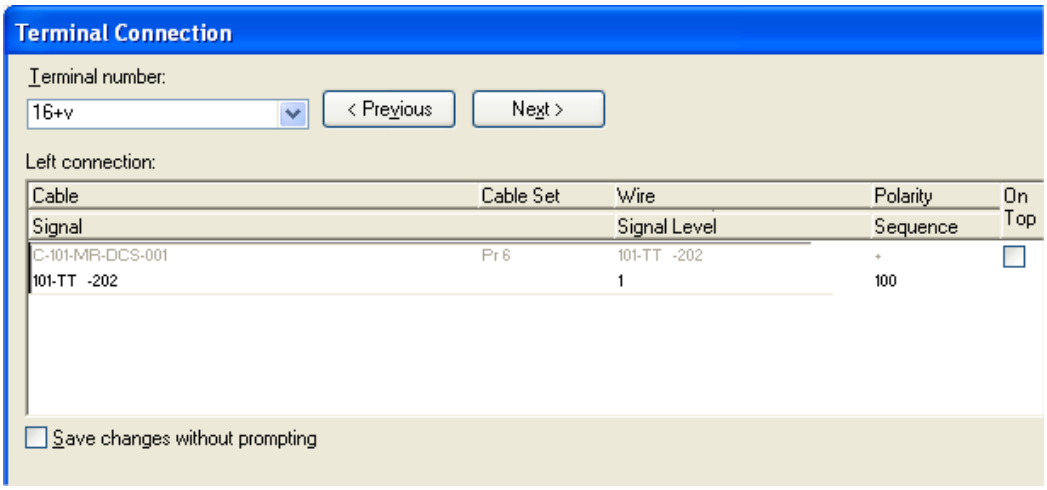
Channel	Address	Control System Tag	Tag Number
1	*	BFI 100	101-FT -100
2	*	BFI 102	101-FT -102
3	*	BLI 201	101-LT -201
4	*		
5	*		
6	*	BTT 202	101-TT -202
7	*	BPI 201	101-PT -201
8	*		

28. In the **Domain Explorer**, expand the hierarchy of **101-DCS-001**, file **1** to display the **8 CH AI** card and its **TB 1** terminal strip.

29. Open the **Connection** window for terminal strip **TB 1** and make sure that cable **C-101-MR-DCS-001**, set **Pr 6** is connected to channel **6**, terminals **16+v**, **17in**, and **18-v** (The signal **101-TT-202** should be propagated to the left in the wires connected to these terminals).



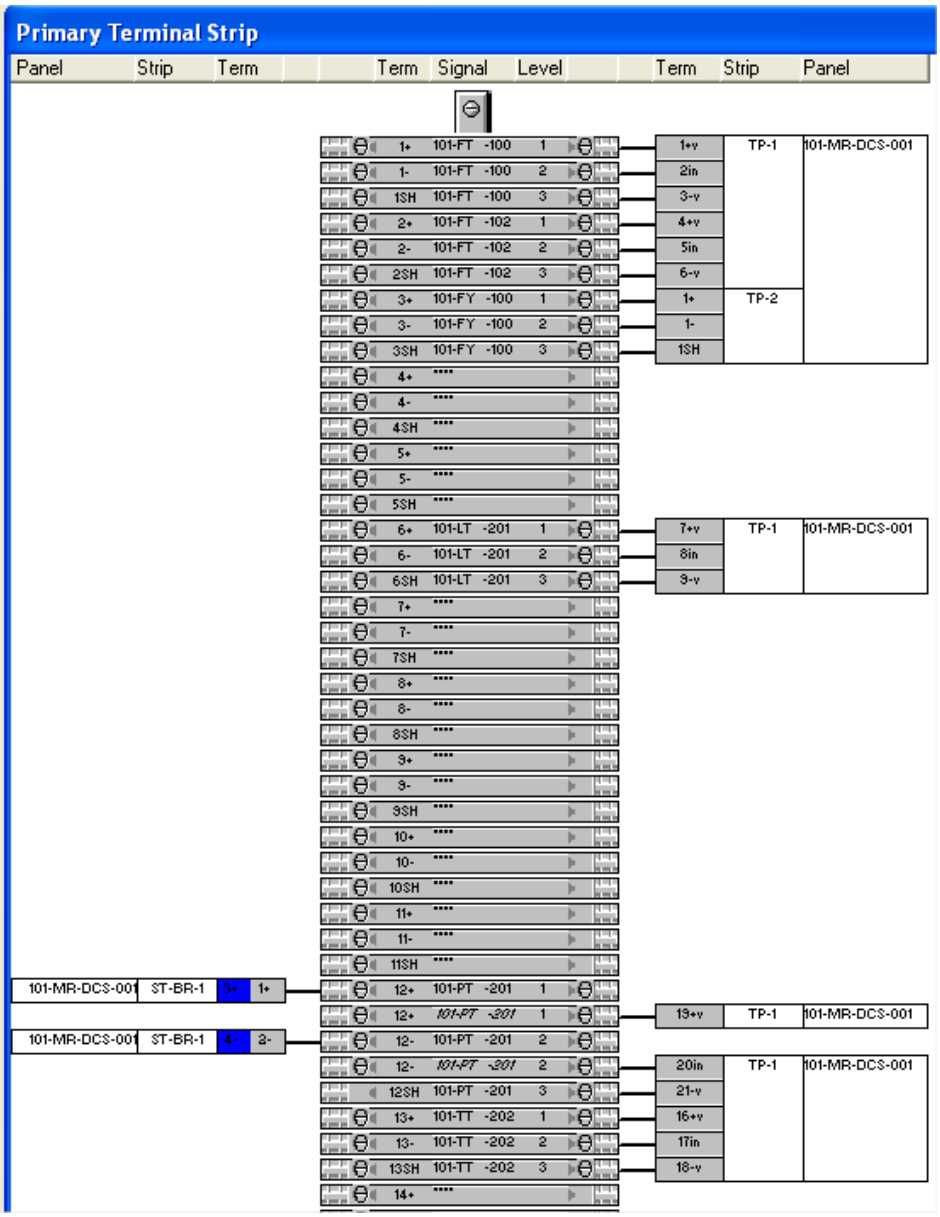
30. Double-click the 16+v screw head to open the **Terminal Connection** dialog box. Make sure that the signal level of this terminal is 1. Then, click **Next** and make sure that the signal level of **17in** is **2** and the signal level of **18–v** is **3** as shown.



31. Select the wire connected to the left of terminal **16+v** and click  to display the connection at the other end of the wire to marshaling rack **101-MR-DCS-001**, strip **TP-1** (This connection was already made in a previous wiring task in this tutorial).
32. Close the **Connection** window for each panel.

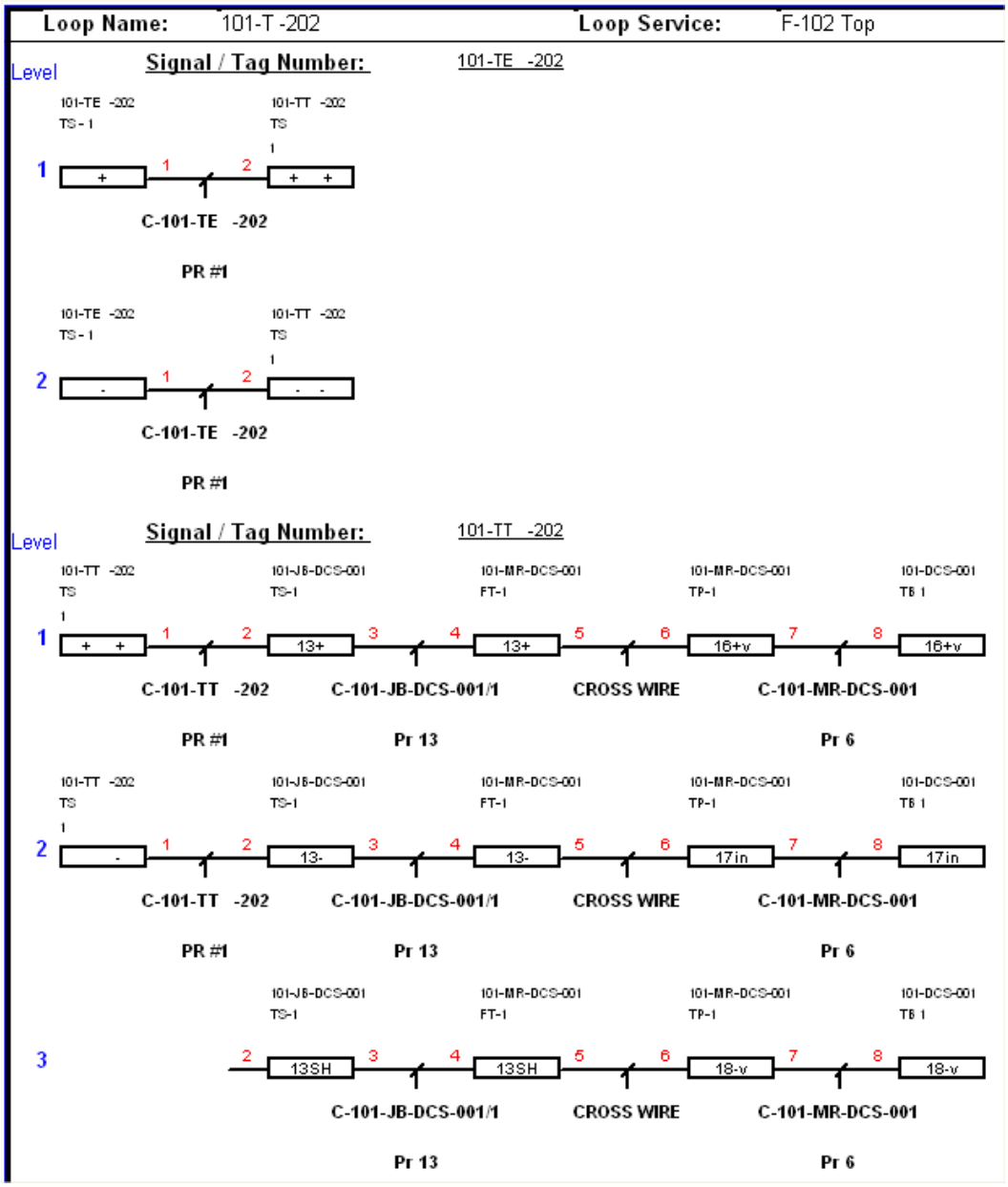
33. In the **Domain Explorer**, select marshaling rack **101-MR-DCS-001** and cross wire terminal strip **FT-1**, terminals **13+** , **13-**, and **13SH** (right side) and terminal strip **TP- 1**, terminals **16+v**, **17in**, and **18-v** (left side) respectively. For details of how to perform cross wiring, see *Task 10 — Cross Wiring the Signals in the Marshaling Rack* (on page 201).

 When done, the cross-wiring connections should be completed as shown:



34. Display the Point-to-Point Wiring Diagram for the loop as follows:
 a. Display the **Connection** window for device panel **101-TT-202** and select any one wire.
 b. Click **Reports** > **Loop Point to Point Diagram**.
 c. When prompted to preview the report, click **Yes**.

The **Point to Point Wiring Diagram** print preview for all the wiring in the loop is displayed. Your connections for Tag Number 101-TT-202 should be the same as shown:

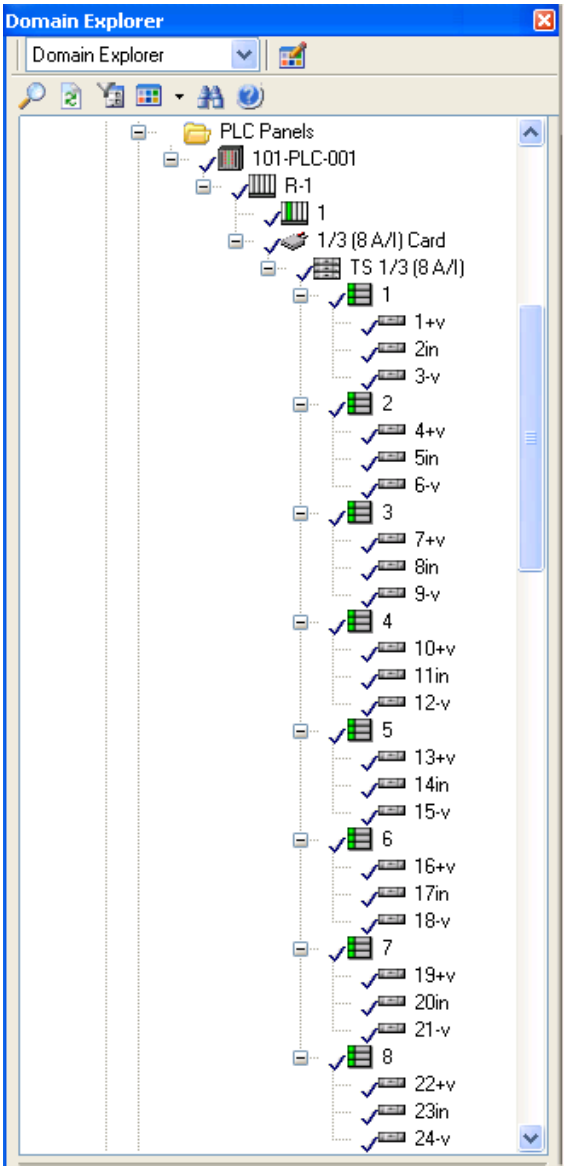


Design the Wiring for Loop F-201

You will now create all the required wiring that constitutes loop **101-F -201**. The wiring for this loop is a little more complicated than for the previous example. In particular, you will learn how to create jumpers and how to adjust a signal path definition manually.

1. In the **Domain Explorer**, create a new junction box **101-JB-PLC-001** by duplicating **101-JB-DCS-001**.
2. In the **Domain Explorer**, create a new multi-cable **C-101-JB-PLC-001** by duplicating cable **C-101-JB-DCS-001/1**.
3. In the **Domain Explorer**, create a new PLC panel and name it **101-PLC-001**.
4. Right-click the new PLC and on the shortcut menu, click **New** > **Rack**.
5. On the **Rack Properties** dialog box, name the rack **R-1** and click **OK**.
6. Right-click the new rack and on the shortcut menu, click **New** > **Slot**.
7. On the **Slot Properties** dialog box, under **Slot**, type **1** and click **OK**.
8. Right-click the rack again, and on the shortcut menu, click **New** > **Wiring Equipment**.
9. On the **New Wiring Equipment** dialog box, select **I/O Card** from the **Category** list and under **Name**, type **1/3 (8 A/I) Card** and click **OK**.
10. On the **Wiring Equipment Properties – I/O Card (New)** dialog box, click the **Control System** tab and from the **I/O type** list, select **AI**.
11. Click **OK** on the **Wiring Equipment Properties – I/O Card (New)** dialog box.
12. In the **Domain Explorer**, right-click the new I/O card and then on the shortcut menu, click **New** > **Terminal Strip**.
13. Create the new terminal strip according to the following definitions:
 a. On the **Terminal Configuration** dialog box, configure the terminal strip to have 8 channels with 3 terminals per channel.
 b. Name the new terminal strip **TS 1/3 (8A/I)**.

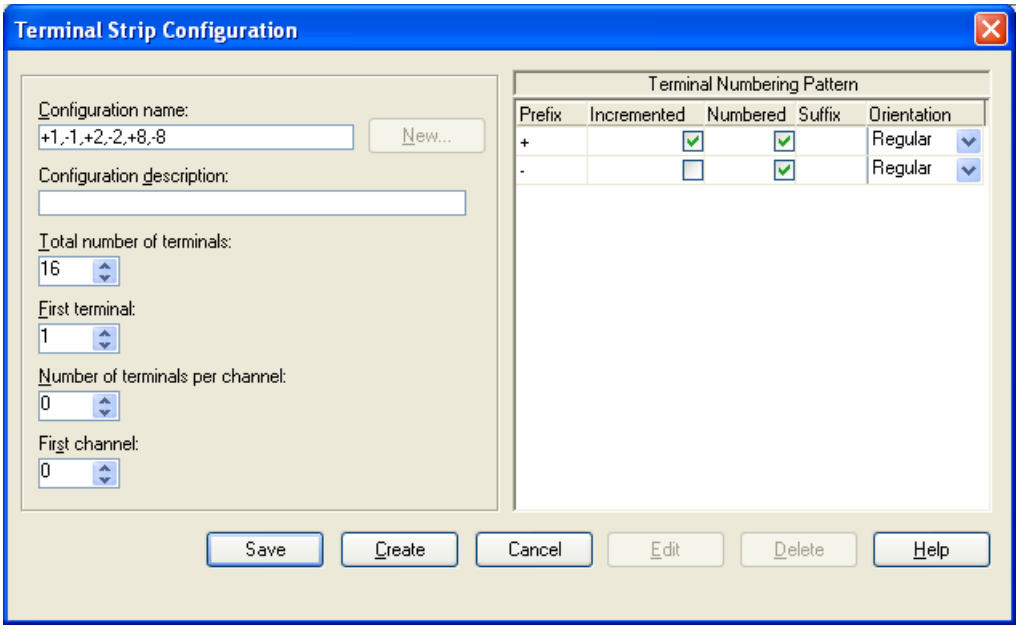
When expanding the hierarchy of the terminal strip in the **Domain Explorer**, the terminals should be numbered as shown:



14. In the **Domain Explorer**, right-click the new I/O card **1/3 (8 A/I) Card**, then on the shortcut menu, click **Actions** and click **I/O Assignment**.
15. In the **I/O Assignment** window, assign tag **101-FT-201** to I/O card **1/3 (8A/I)**, channel **#1**. Name the CS tag **BFT201**.

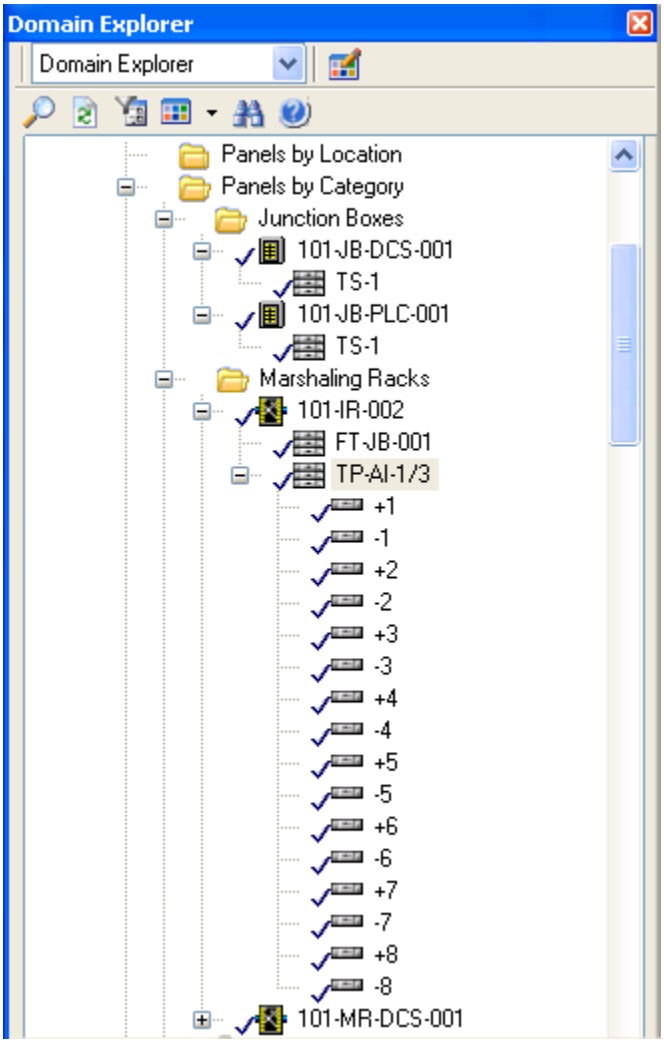
 TIP If Tag 101-FT-201 is not on the list in the **Tag List** pane of the **I/O Assignment** window, click  and select the **Uncoupled tags** check box.
16. Create a new marshaling rack and name it **101-IR-002**.
17. Add a new terminal strip for the field side of **101-IR-002** using the following definitions:
 a. Using the **2 TERMINALS+SHIELD** configuration, configure the strip to have 16 blocks of terminals with 3 terminals per block.
 b. Name the new strip **FT-JB-001**.

18. Add a new terminal strip for the PLC side of **101-IR-002** using the following definitions:
 a. Create a new terminal strip configuration as shown.



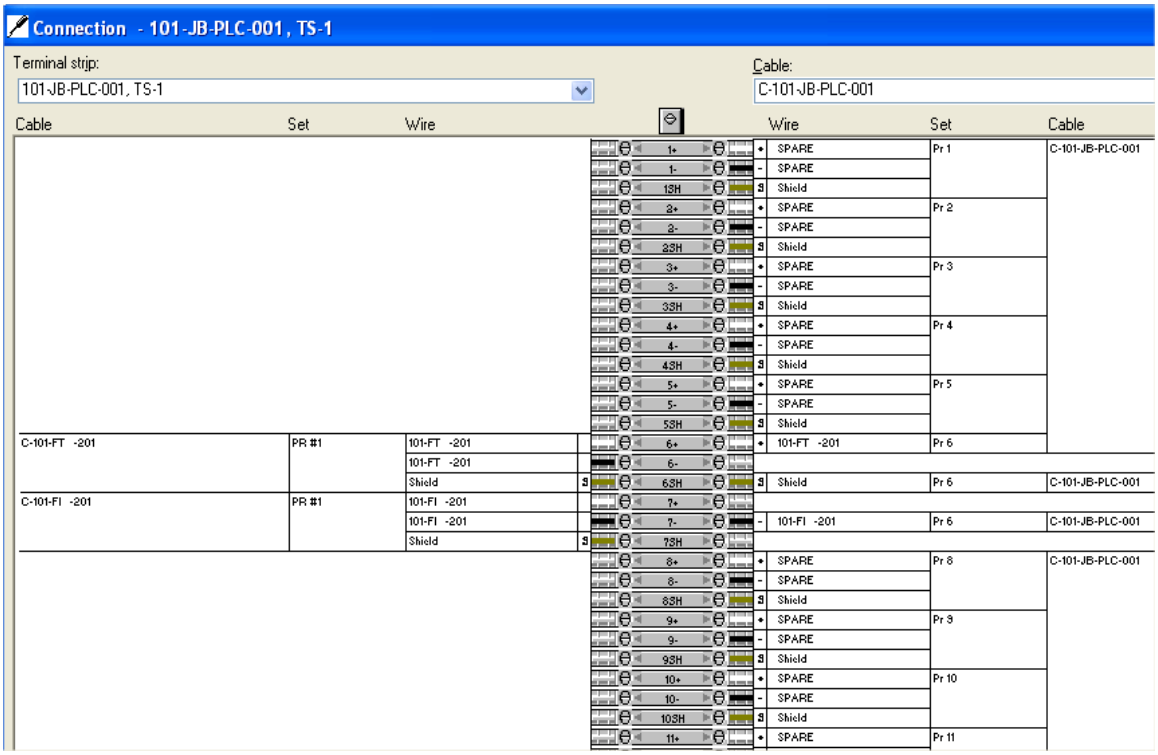
19. Create a new terminal strip and name it **TP-AI-1/3**.

20. Expand the terminal strip in the **Domain Explorer** and ensure that the terminals are numbered as shown.



21. Make the following connections in junction box **101-JB-PLC-001**, strip **TS-1**:
 a. Connect cable **C-101-FT-201** starting at terminal **6+** at the left side of the terminal strip using **3 in a row** as the connection type.
 b. Connect cable **C-101-FI-201** starting at terminal **7+** at the left side of the terminal strip using **3 in a row** as the connection type.
 c. Connect multi-cable **C-101-JB-PLC-001** starting at terminal **1+** at the right side of the terminal strip. Connect all the cable sets on the JB cable side using **3 in a row** as the connection type.
22. View the connections at terminals **6+** to **7-** and note the signal propagation in the wires at the right side of the terminal.
23. Select cable set **Pr 7** with wires connected on the right side of the strip and disconnect this set.

24. Select the wire connected to the right side of terminal **6-** and drag it to the right side of terminal **7-**.

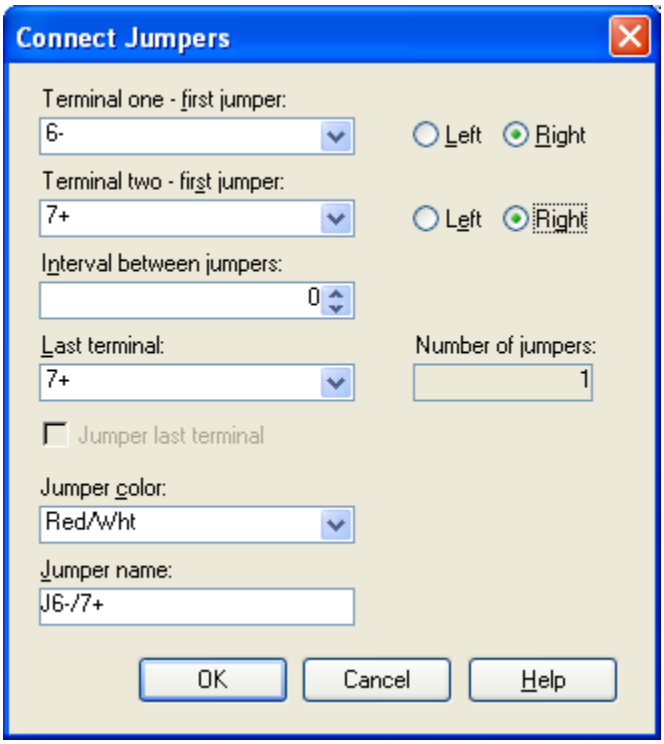


Note that the signal propagation changes for this wire after reconnection. You will later change the signal definition for this wire manually.

25. Add a jumper between terminals **6-** and **7+** on the right side of the terminal strip as follows:

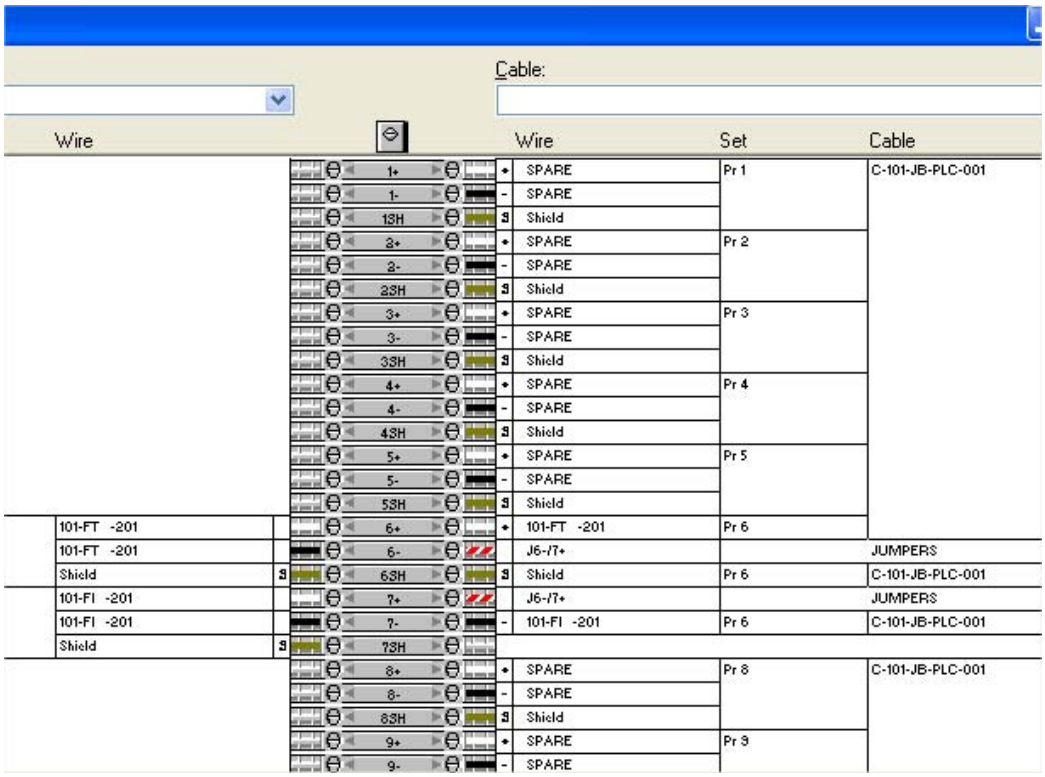
 a. Click .

b. On the **Connect Jumpers** dialog box, enter values as shown.

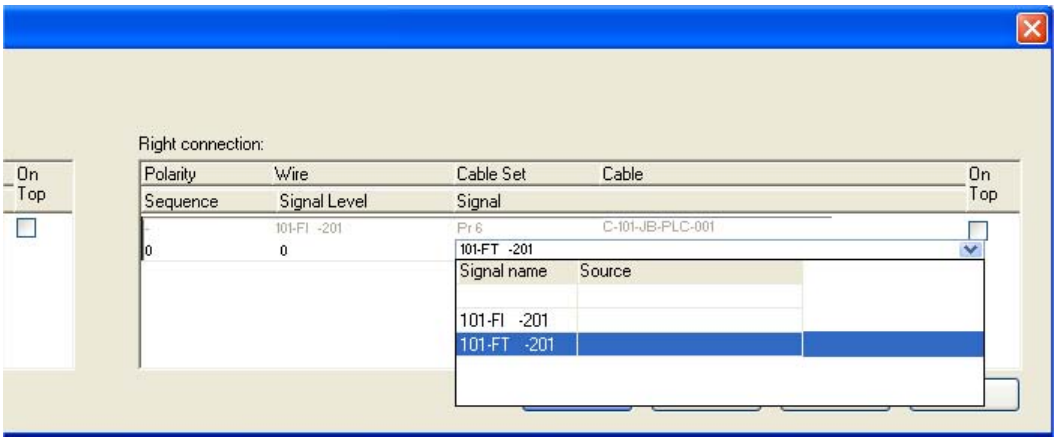


c. Click **OK**.

The connections should appear as shown:

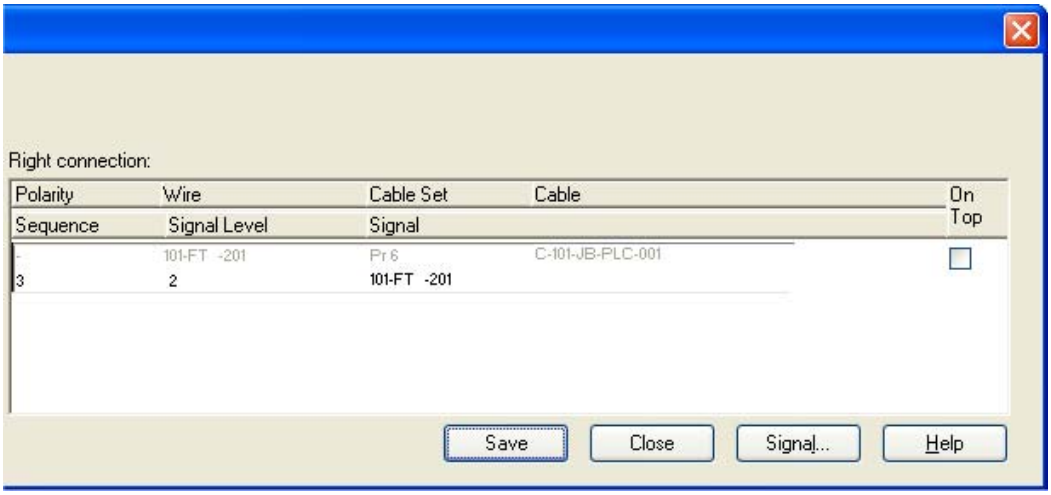


26. Change the signal definition as follows for the wire you have moved:
 a. Double-click a screw head on terminal **7-** to open the **Terminal Connection** dialog box.
 b. From the **Signal** list on the right side, select **101-FT-201**.



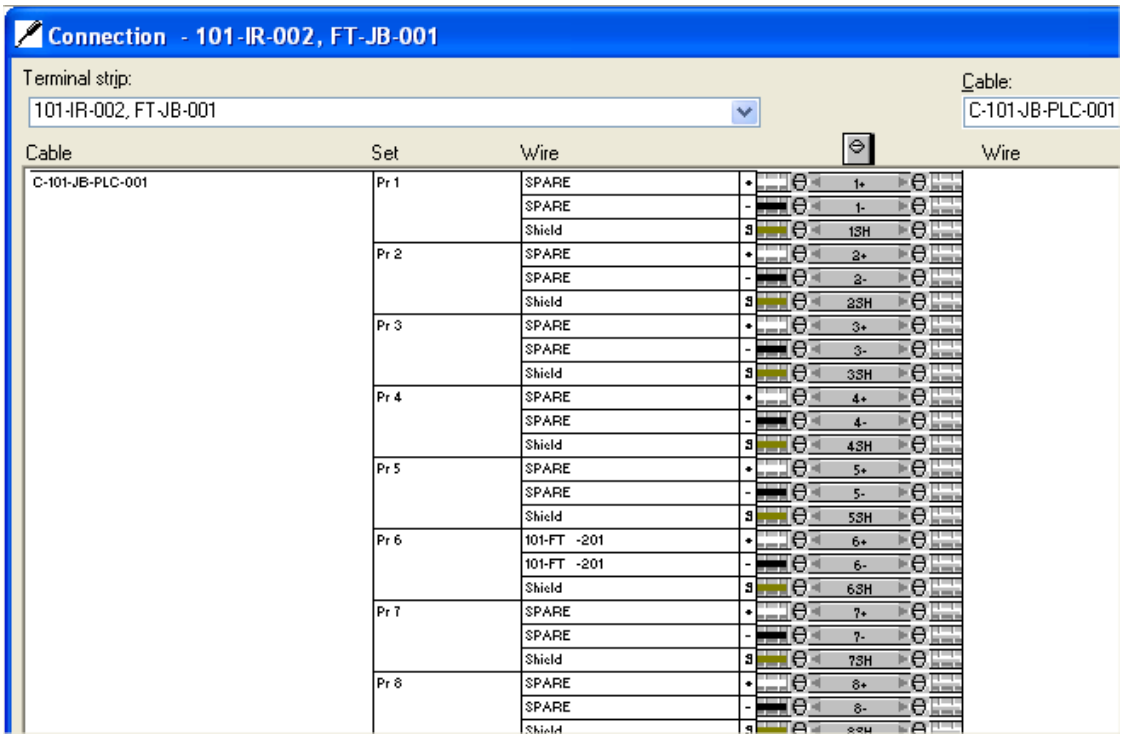
 c. From the **Signal Level** list on the right side, select 2.
 d. From the **Sequence** list on the right side, select 3.
 e. Click **Save**.

 The right side of the **Terminal Connection** dialog box should appear as shown:



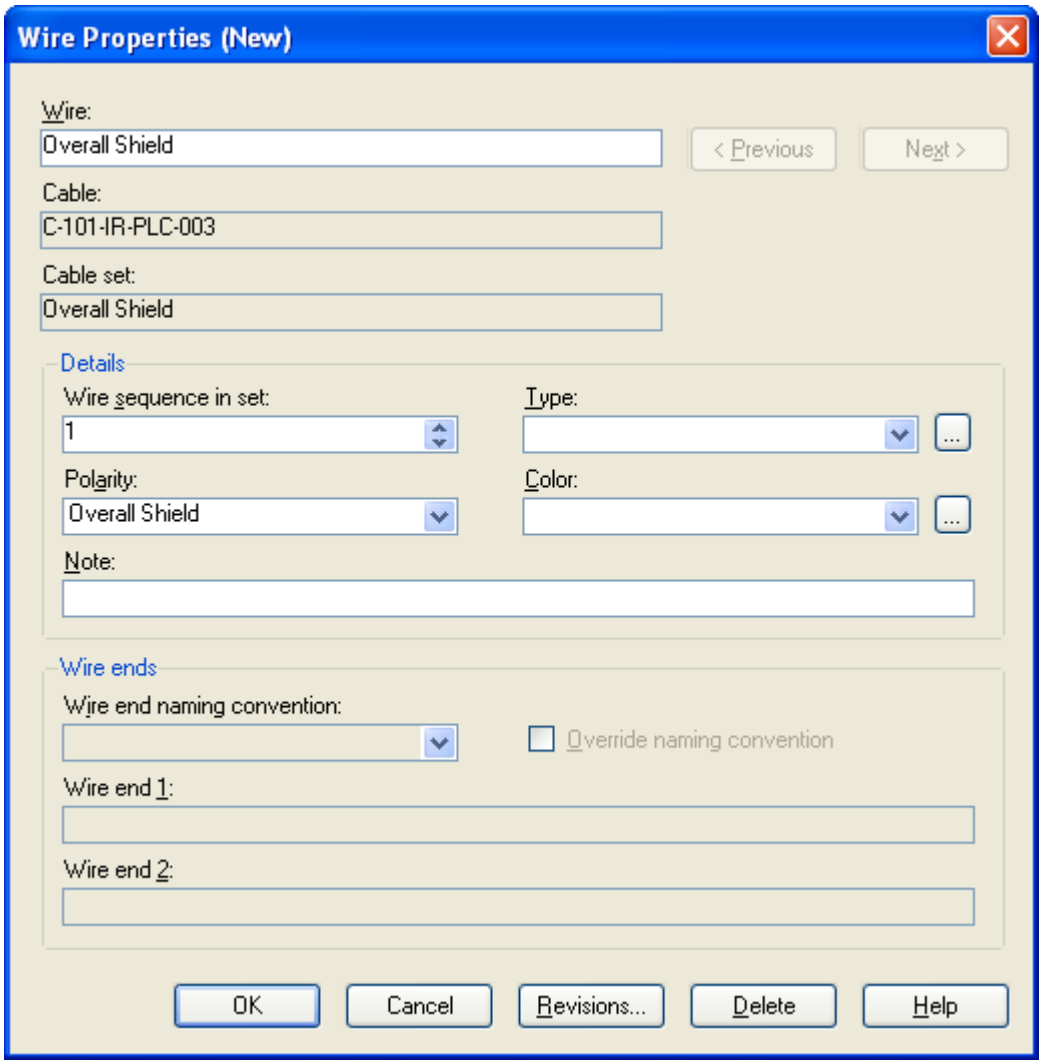
 f. Click **Close** on the **Terminal Connection** dialog box.
27. Close the **Connection** window and then reopen it for marshaling rack **101-IR-002**, terminal strip **FT-JB-001**.
28. Connect the other end of cable **C-101-JB-PLC-001** to terminal **1+** at the left side. Connect all the cable sets on the **MARSHALING** cable side using **3 in a row** as the connection type.
29. On the **Connect Options** dialog box, click **OK**.

30. View the connections at terminals **6+** to **7SH**.



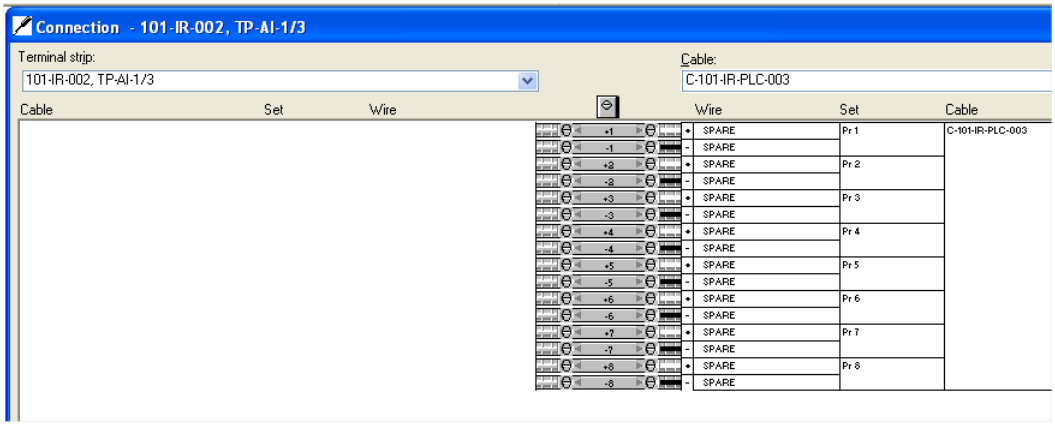
31. When done, close the **Connection** window.
32. In the **Domain Explorer**, create cable **C-101-IR-PLC-003** by duplicating cable **C-101-MR-DCS-001**. Rename **End 1 Location** as **Instrument Rack** and **End 2 Location** as **PLC**.
33. Add an overall shield to the cable as follows:
 a. Select the cable and add a new cable set to it named **Overall Shield**.
 b. Select the new cable set and add a new cable wire to it.
 c. Under **Wire**, type **Overall Shield**.

d. Under **Polarity**, select **Overall Shield**.



34. Connect cable **C-101-IR-PLC-003** between marshaling rack **101-IR-002**, terminal strip **TP-AI 1/3** and **PLC 101-PLC-001**, terminal strip **1/3 (8A/I)** with connection definitions as follows:
 a. Open the **Connection** window for marshaling rack **101-IR-002**, terminal strip **TP-AI 1/3**.
 b. Connect cable **C-101-IR-PLC-003** to terminal **1+** at the right side. Connect all the cable sets **except** for **OVERALL SHIELD**, using **2 in a row** as the connection type.

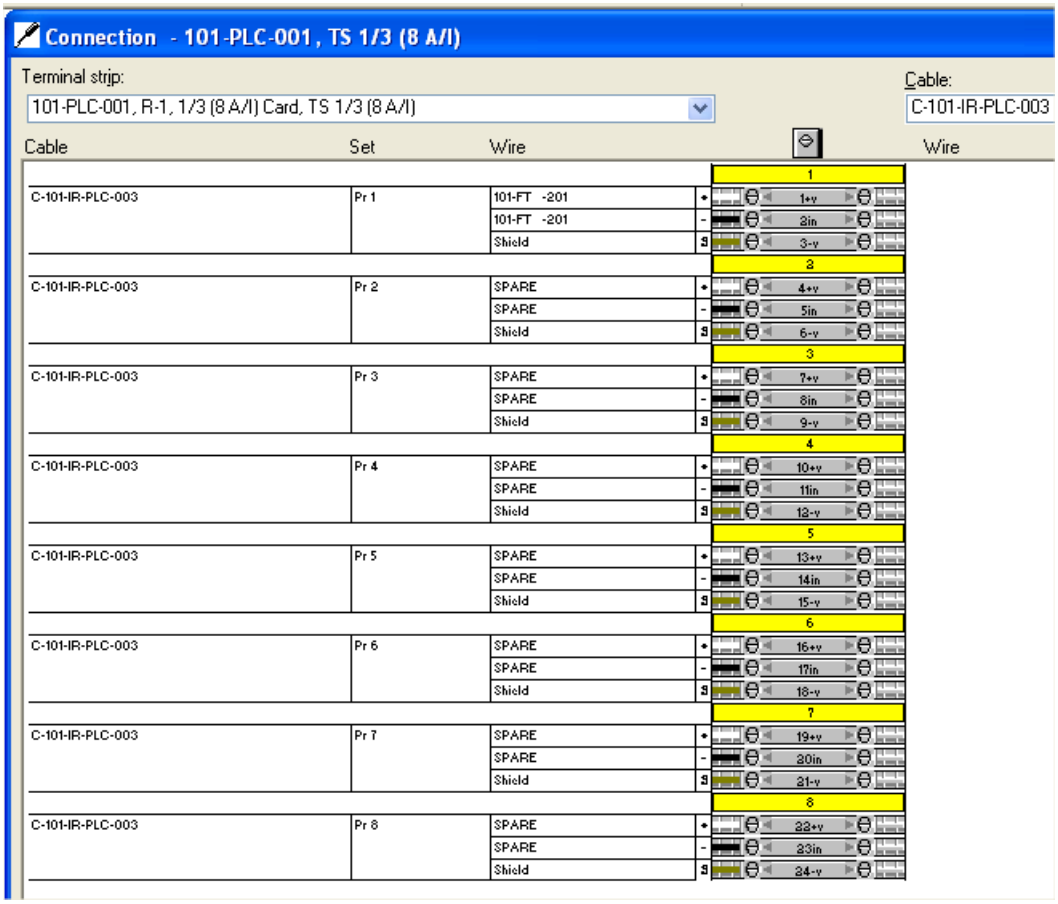
The connection should appear as follows:



c. Open the **Connection** window for PLC panel **101-PLC-001**, terminal strip **TS 1/3 (8A/I)**.

d. Connect cable **C-101-IR-PLC-003** to terminal **1+** at the left side. Connect all the cable sets **except** for **OVERALL SHIELD**, using **3 in a row** as the connection type.

The connection should appear as shown:

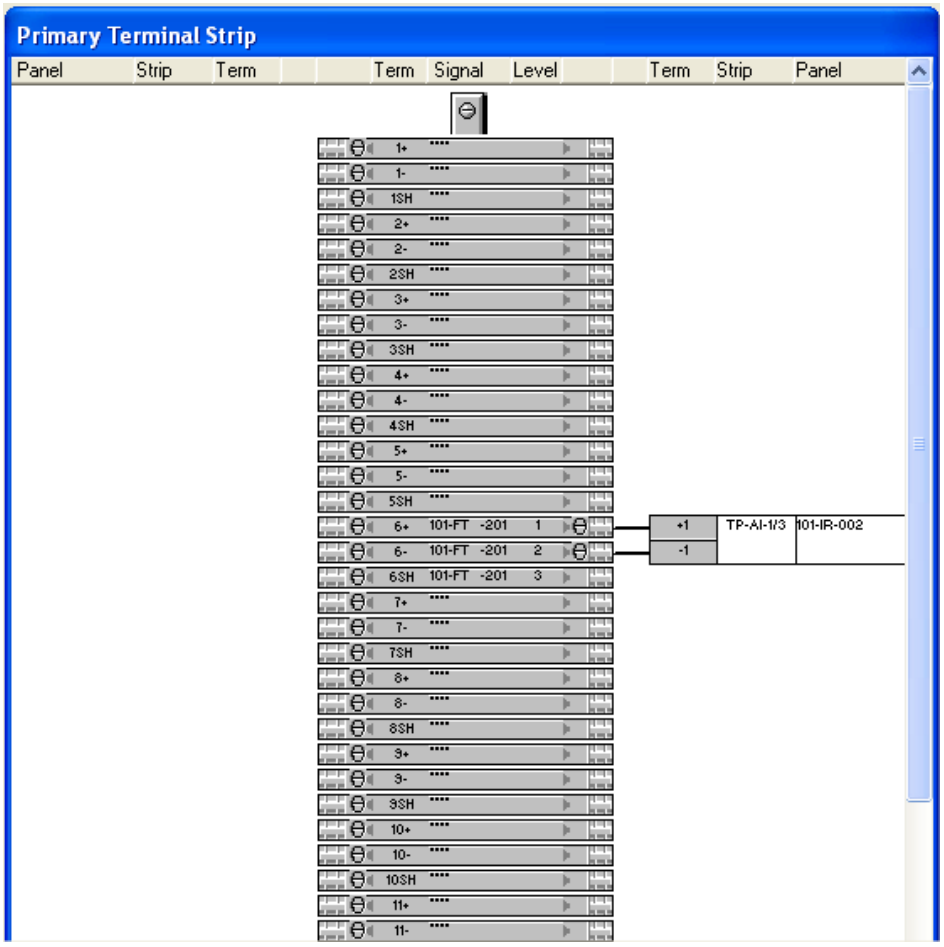


NOTE A signal for tag **101-FT-201** appears for the first two wires because of the I/O assignment you made in step 15.

e. Disconnect the shield wire connected to terminal **3-v**.

35. When done, close the **Connection** window.
36. Select marshaling rack **101-IR-002** and cross-wire terminal strip **FT-JB-001**, terminals **6+** and **6-** (right side) and terminal strip **TP-AI-1/3**, terminals **+1** and **-1** (left side) respectively.

 When done, the cross-wiring connections should be completed as shown:



This completes the wiring design for loop 101-F -201.

This completes all the wiring required for this tutorial. We would like to draw your attention to the fact that some functions of the Wiring module were omitted because the examples in the Tutorial do not use them; nevertheless, you are encouraged to familiarize yourself with the following functions:

- **Connection Type** can be defined by clicking . Simple and complex versions of connection types can be created in a plant related list and reused as required for further automation of the wiring design. For details, read the **Define a Connection Type** section in the **SmartPlant Instrumentation Detailed Engineering Users Guide** document.
- **General Signals and Tag Signals** allow the creation of signals that do not originate from field devices. **General Signal** refers to a signal that may be associated with several tags, for example, a common power supply. **Tag Signal** is a unique signal, for example, the digital signal from a cabinet mounted trip amplifier. It will require this device to be listed in the Instrument Index and its profile will have the wiring checkbox selected, however, without defining a device panel or an instrument cable.

SECTION 11

Generating Loop Drawings

This section explains how to generate loop drawings using the Enhanced Report Utility that is available with your SmartPlant Instrumentation installation. To install the Enhanced Report Utility, On the SmartPlant Instrumentation CD Browser, click **Enhanced Report Utility Installation**.

You can generate loop drawings using the data that you saved to the database when you designed your wiring.

Generating Loop Drawings Using the Enhanced Report Utility

This section explains how to generate loop drawings using the Enhanced Report Utility that is available with your SmartPlant Instrumentation installation. To install the Enhanced Report Utility, On the SmartPlant Instrumentation CD Browser, click **Enhanced Report Utility Installation**.

You can generate loop drawings using the data that you saved to the database when you designed your wiring.

Task 1 — Setting Preferences

Before generating reports in the Enhanced Report Utility you must set the preferences for your drawings in SmartPlant Instrumentation. These preferences set how your SmartLoop drawings are displayed in the Enhanced Report Utility.

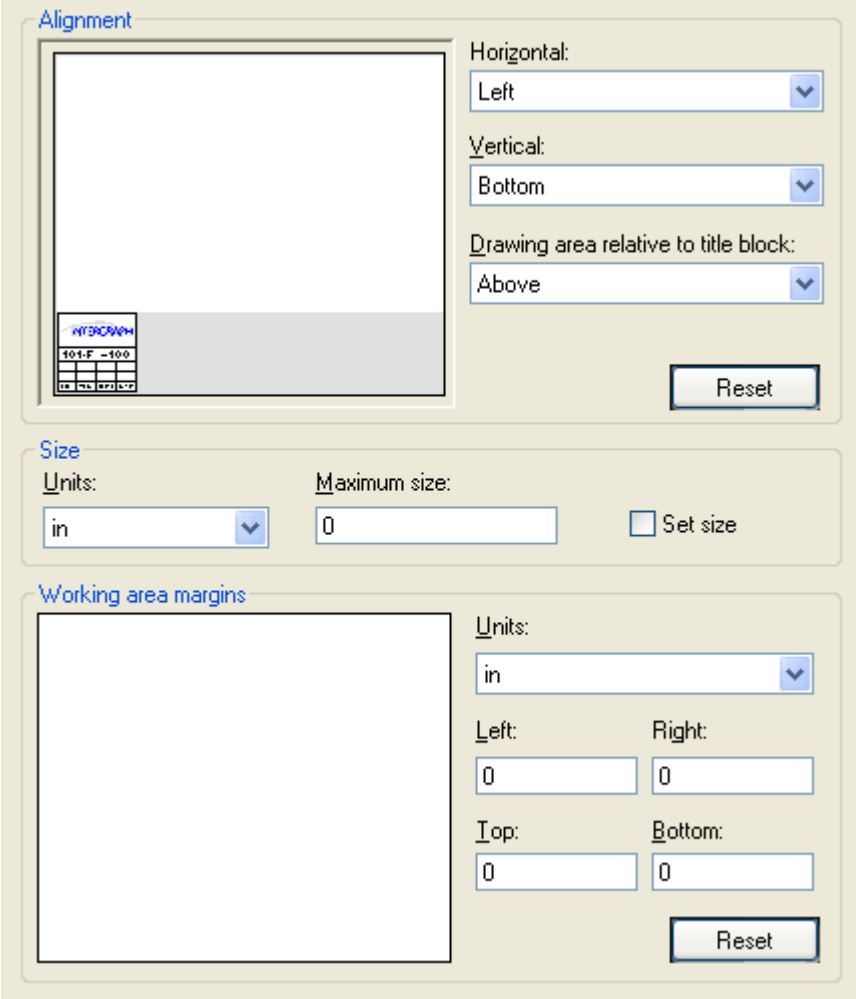
Set Preferences for Enhanced SmartLoop Reports

1. Click **File** > **Preferences**.
2. In the tree, navigate to **Loop Drawings** > **General**.
3. Under **Default generation method**, select **Enhanced Report**.

 TIP You can override the default generation method for each loop individually prior to generating the loop drawing or report.
4. In the tree, navigate to **Enhanced Reports** > **Enhanced SmartLoop**.
5. Expand the tree and click **View**.
6. Under **Display options**, select the desired settings.

 TIP Click **Help** to display the Online Help with an explanation of each setting.
7. In the tree, click **Drawing**.
8. Under **Generation method**, click **By loop**.
9. In the tree, click **Title Block**.

10. Ensure that the settings appear as shown.



11. Click **OK**.

Apply Generation Method to Individual Loops

1. In the **Domain Explorer**, expand the plant hierarchy **New Refinery** > **Crude Area** > **Crude Unit 1** > **Loops**, and select loop **101-F –100**.

 TIP For details of how to locate an item using the **Search** command, see steps 1 and 2 of the topic *Search for a Specific Loop and View Loop Tag Properties* (on page 80).

2. Right-click the loop, and on the shortcut menu, click **Apply Generation Method** > **Enhanced Report** > **By Loop**.

 The symbol **E** appears to the left of the loop icon for loop **101-F -100**.

3. Now select and then right-click loop **101-F -102**.
4. On the shortcut menu, click **Apply Generation Method** > **Enhanced Report** > **By Signal**.

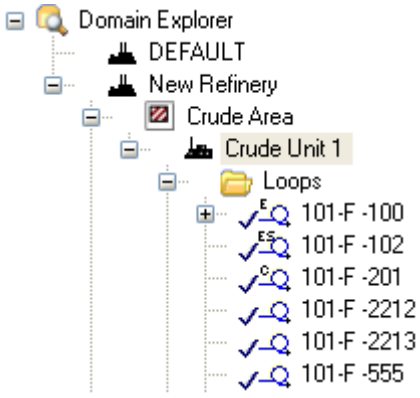
 The symbol **ES** appears to the left of the loop icon for loop **101-F -102**.

5. Now select and then right-click loop **101-F -201**.
6. On the shortcut menu, click **Apply Generation Method** > **CAD**.

 The symbol **C** appears to the left of the loop icon for loop **101-F -201**.

TIP You can apply the same indication to several loops by holding down **Shift** or **Ctrl** while selecting the required loops in the **Domain Explorer** and then selecting the appropriate generation type.

The **Domain Explorer** should appear as shown:



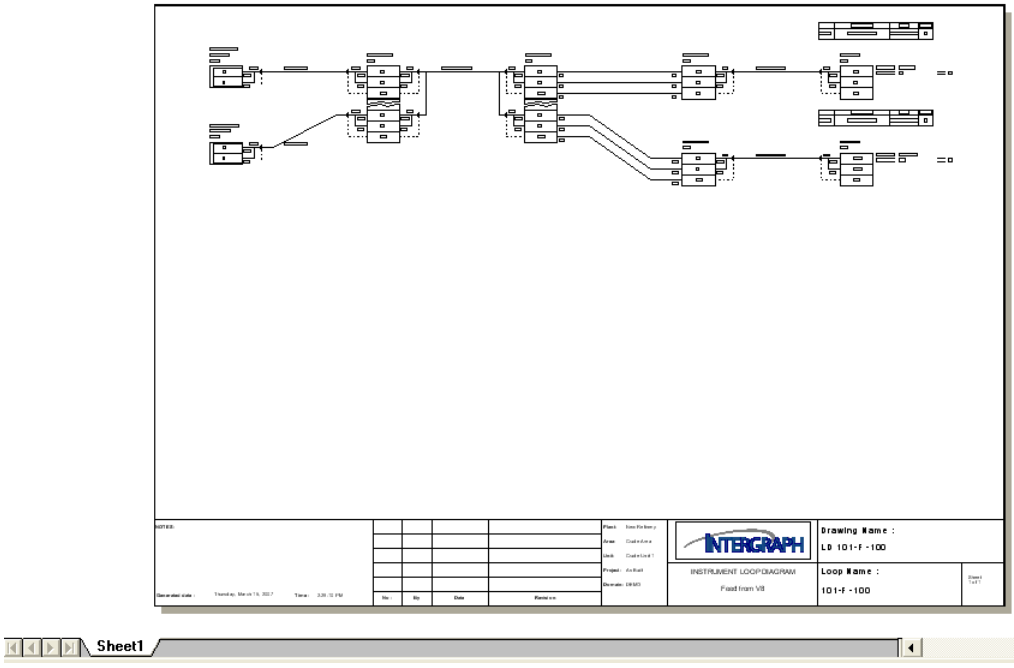
Task 2 — Generating an Enhanced SmartLoop Drawing

In this task you will generate an Enhanced SmartLoop drawing using the wiring information that you have already created.

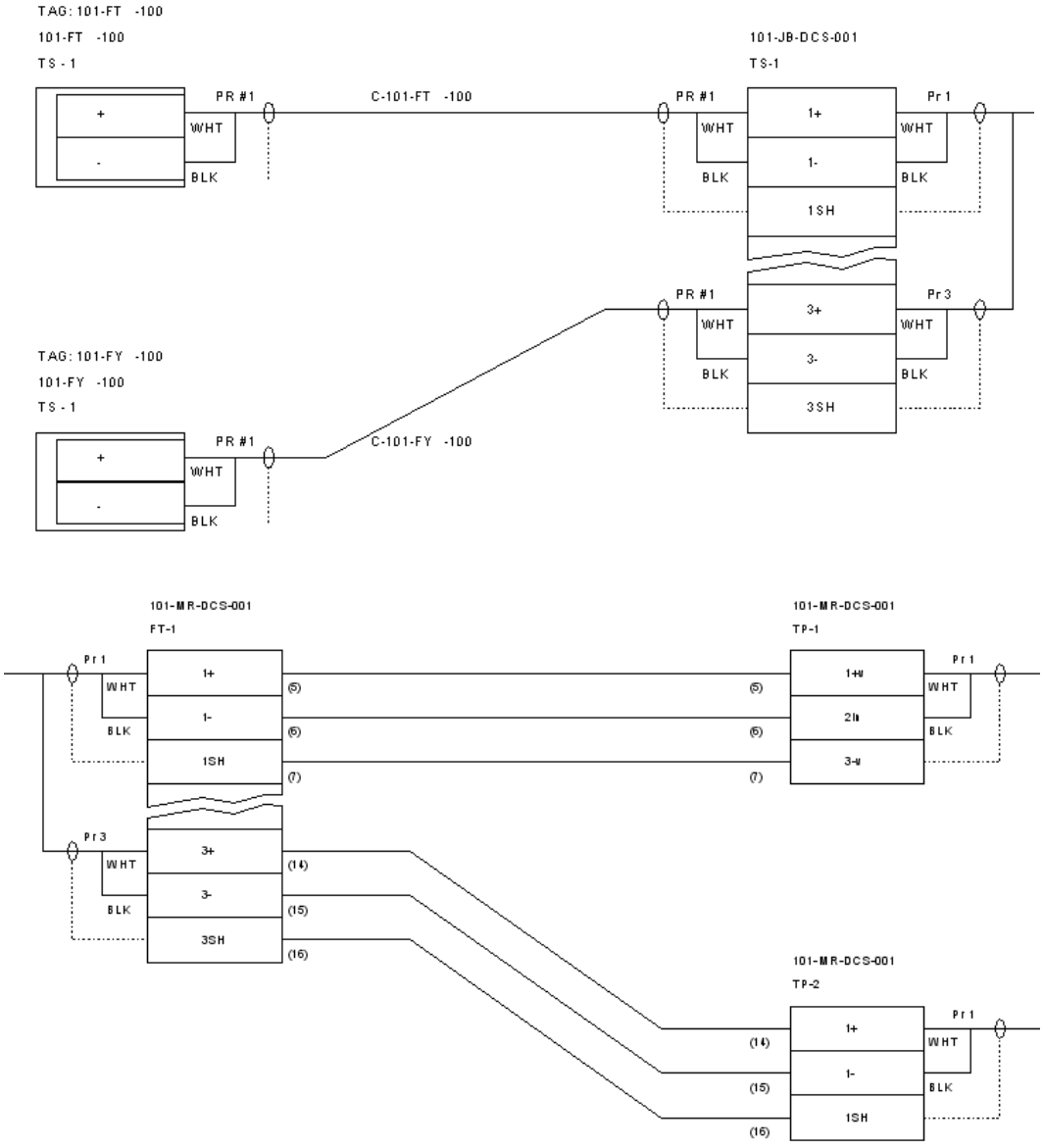
Generate an Enhanced SmartLoop Drawing

1. In the **Domain Explorer**, select and then right-click loop **101-F -100**.
2. On the shortcut menu, click **Reports** > **Generate Loop Drawings**.
3. On the **Generate Loop Drawings** dialog box, make sure that the **Generation method** list displays **Enhanced Report**, and then click **OK**.
4. When prompted to preview the report, click **Yes**.

A **Progress** window appears and the Enhanced Report Utility opens with the report displayed:

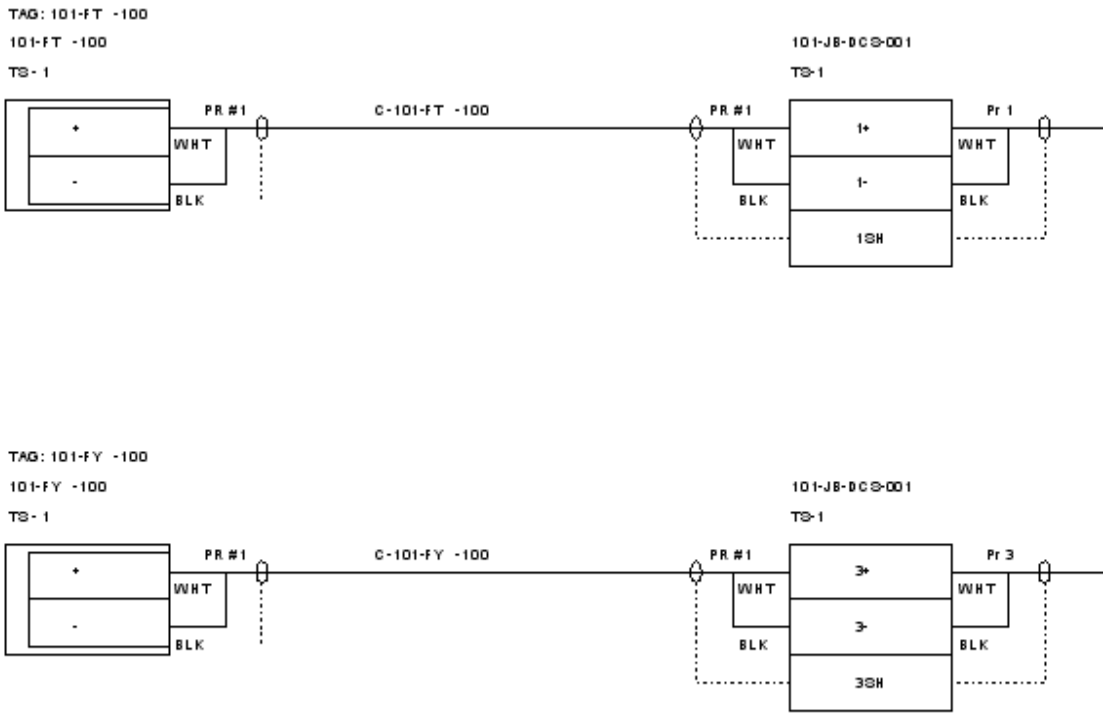


5. On the **Main** toolbar, click the **Zoom In** command and drag over portions of the drawing to zoom in and display more details.



Change the Loop Generation Method to Show Individual Signals

1. In the **Domain Explorer**, select and then right-click loop **101-F -100**.
2. On the shortcut menu, click **Apply Generation Method** > **Enhanced Report** > **By Signal**.
 The symbol **ES** appears to the left of the loop icon for loop **101-F -100**.
3. In the Enhanced Report Utility, on the **View** toolbar, click **Refresh** ().
 The Enhanced Report Utility refreshes the report and displays each signal separately as shown:

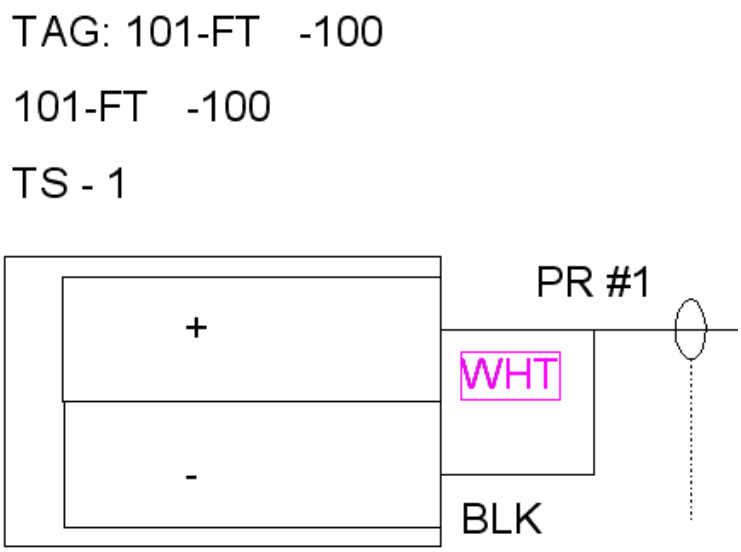


4. Change the generation method for loop **101-F -100** back to **Enhanced Report** > **By Loop**.

Task 3 — Modifying an Enhanced SmartLoop Drawing

Modify Properties of Items from the Enhanced Report Utility

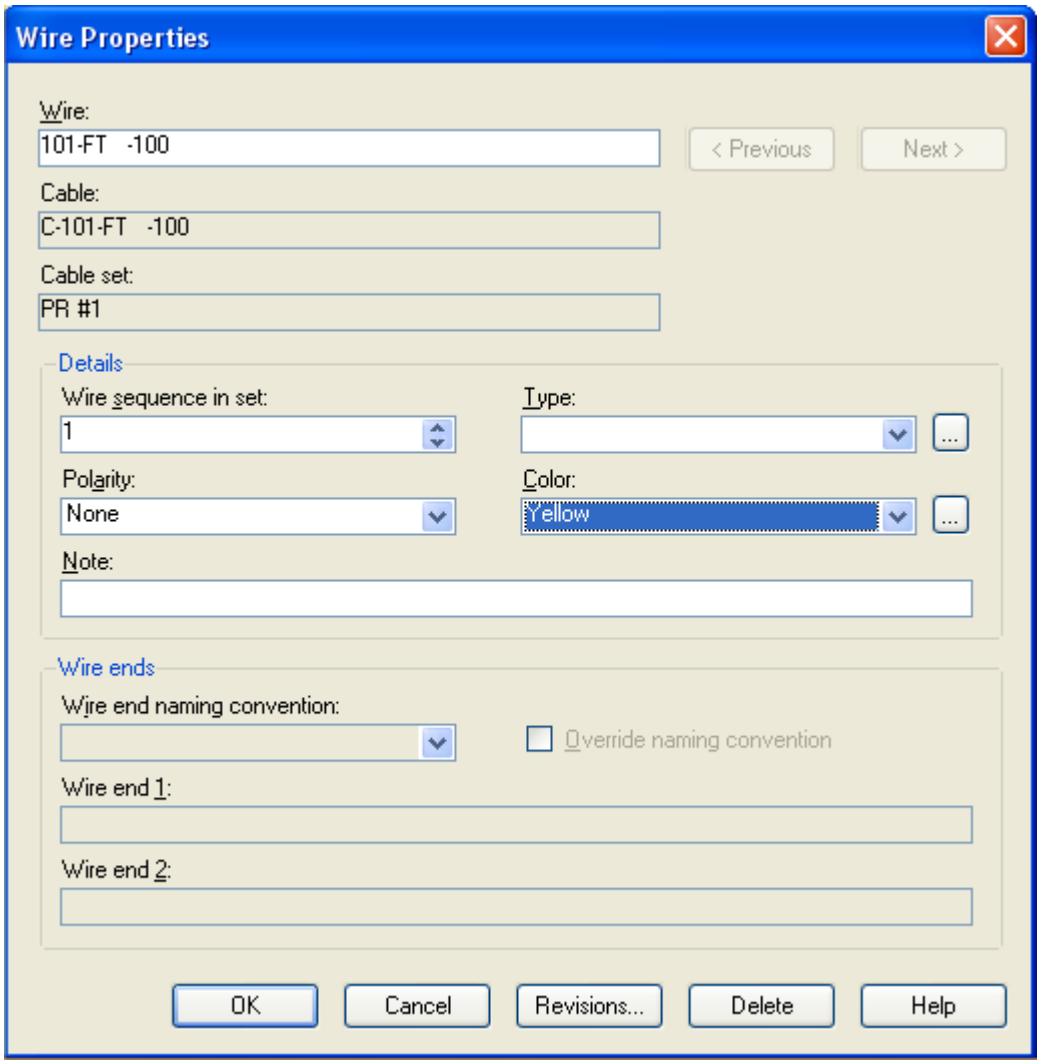
1. In the **Enhanced Report Utility,** select the upper wire that is connected to the device panel for tag **101-FT -100**, as shown.



2. Do one of the following:
 - On the **Actions** toolbar, click .
 - Click **Actions** > **Item** > **Properties**.

 TIP If not done automatically, switch back to **SmartPlant Instrumentation**.

3. On the **Wire Properties** dialog box, from the **Color** list, select **Yellow**.



4. Click **OK** to return to the Enhanced SmartLoop report.

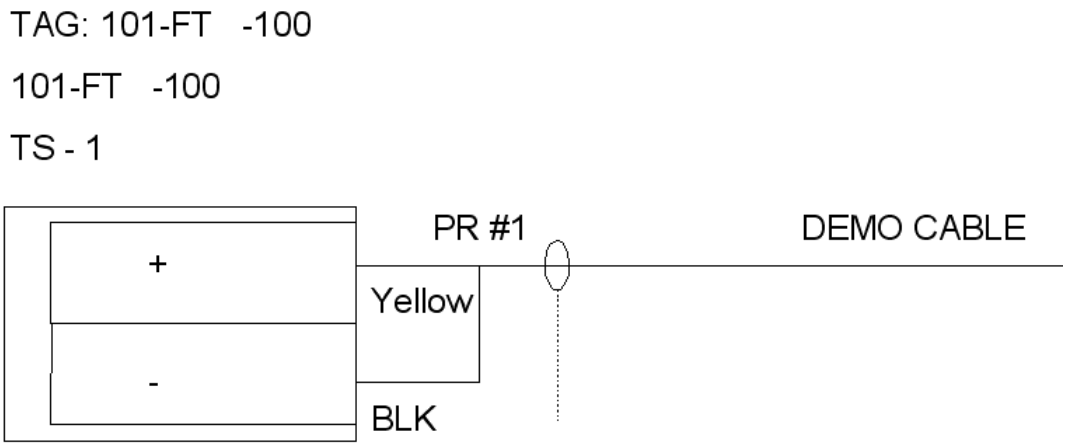
 TIP The wire color value is not updated at this stage.

5. Now position the cursor over cable **C-101-FT -100** and repeat the previous steps to change the cable name to **DEMO CABLE**.

6. To update the modified data values, on the **Actions** toolbar, click **Refresh** ().

 TIP If the **Refresh** command is not enabled, click a blank region of the drawing sheet first.

The Enhanced Report Utility regenerates the loop drawing with the updated data values:



7. Restore the properties that you changed to their original values and refresh the report again.

Open Documents Associated with Instruments

It is possible to open a document associated with a specific instrument directly from the generated loop drawing.

1. On the currently open drawing page of the Enhanced Report Utility, select the **TAG: 101-FT -100** label.
2. On the **Actions** toolbar, click **Tag Documents** ().
3. On the **Documents** window that opens in SmartPlant Instrumentation, click **Specification** to open the flow transmitter specification.
4. Click to close the specification sheet.

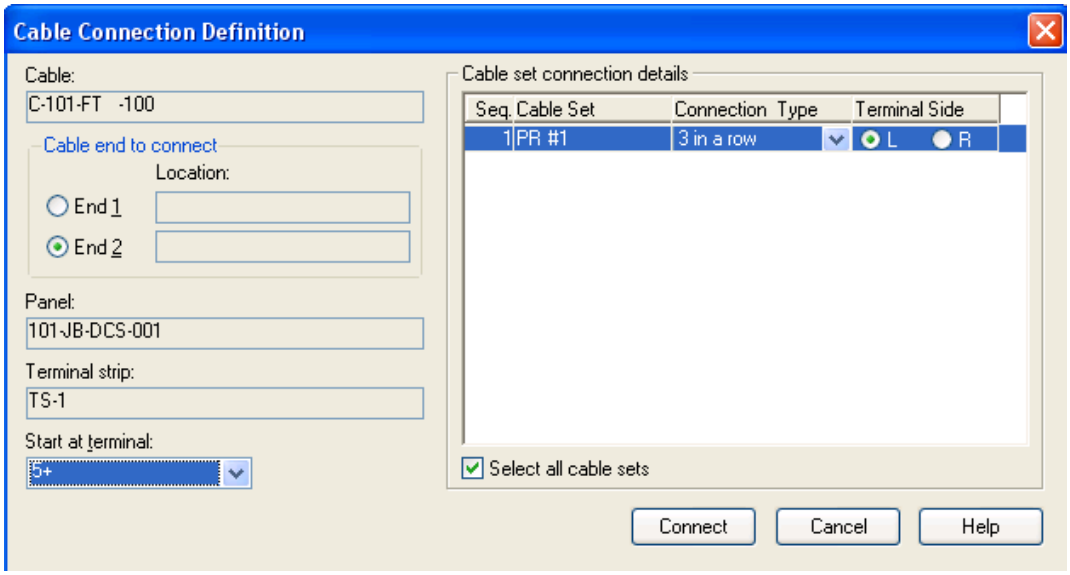
Modify the Wiring Connection from the Enhanced Report Utility

Suppose that in loop 101-F -100, you want to move the connections in junction box 101-JB-DCS-001 from terminals 1+ and 1- to terminals 5+ and 5-.

1. In the **Domain Explorer**, select and then right-click loop **101-F -100**.
2. On the shortcut menu, click **Reports** > **Generate Loop Drawings**.
3. On the **Generate Loop Drawings** dialog box, make sure that the **Generation method** list displays **Enhanced Report**, and then click **OK**.
4. At the print preview prompt, click **Yes**.
5. Click and zoom in on junction box **101-JB-DCS-001**.
6. Position the cursor over terminal strip **TS-1** of junction box **101-JB-DCS-001** and select the label **TS-1**.
7. On the **Actions** toolbar, click **Connection Window** ().
8. In the **Connection** window, select cable **C-101-FT-100** (on the left side of the terminal strip) and drag it to terminal **5+**.

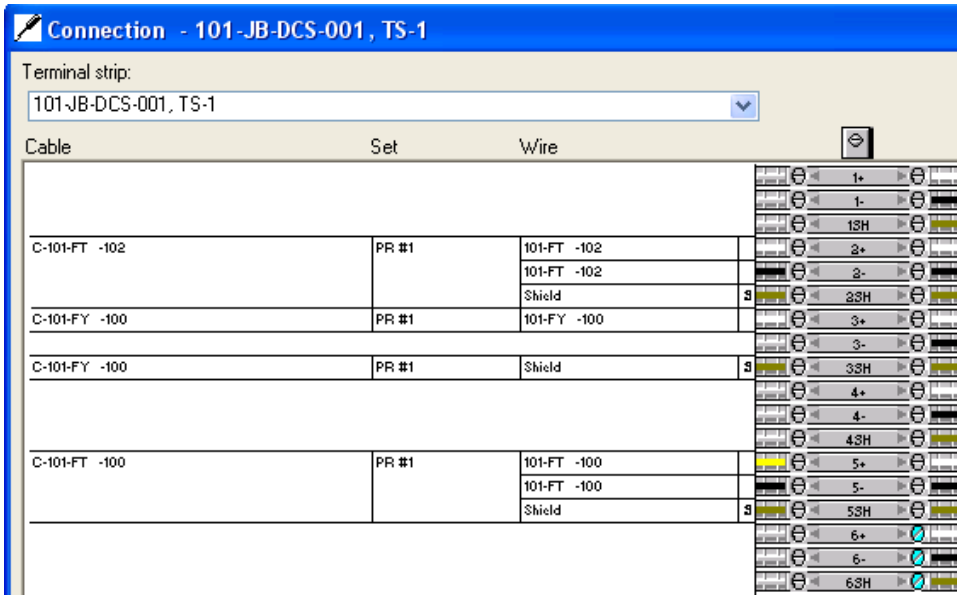
 TIP The window scrolls automatically if you drag the cable to the lower edge.

9. On the **Cable Connection Definition** dialog box, make your cable connection definitions as shown.



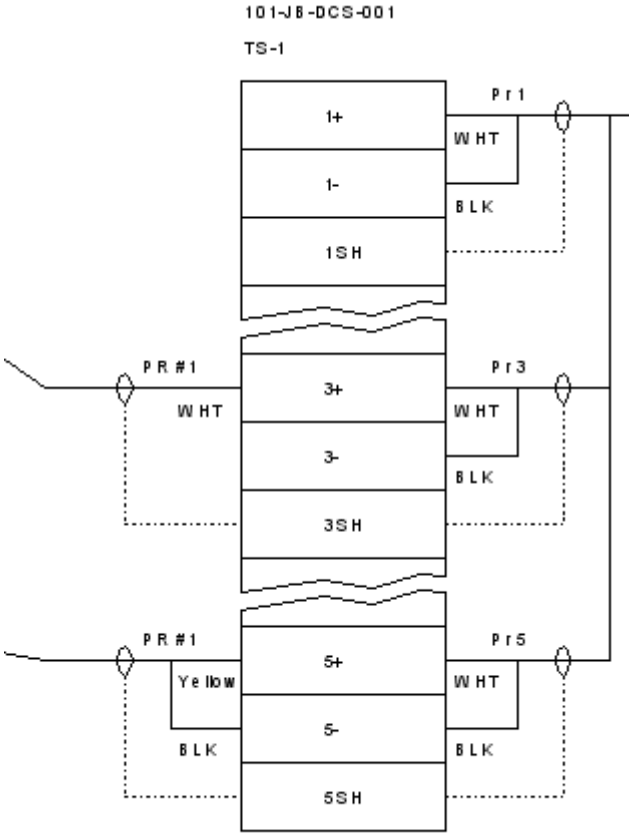
10. Click **Connect** to make the connection.

 The cable moves to its new position (with wires on terminals **5+**, **5-**, and **5sh**) as shown:



11. Click  to close the **Connection** window.
12. Return to the Enhanced Report Utility.

13. On the **Actions** toolbar, click  to refresh the report and display the new connections.



TIPS

- If the **Refresh** command is not enabled, click a blank region of the drawing sheet first.
- If, after clicking the **Refresh** command, the data fails to update, close and then reopen the Enhanced Report Utility.

14. On the **File** menu, click **Save As**, and on the **Save As** dialog box, do the following:
 a. Type a name for the file and navigate to the folder where you want to save the file.
 b. In the **Save as type** list, select **AutoCAD (*.dxf)** and click **Save**. You can import the file saved in this format into SmartSketch, AutoCAD or MicroStation.
15. Move the cable back to its original position and regenerate the Enhanced SmartLoop drawing.
16. Click  to close the Enhanced Report Utility.

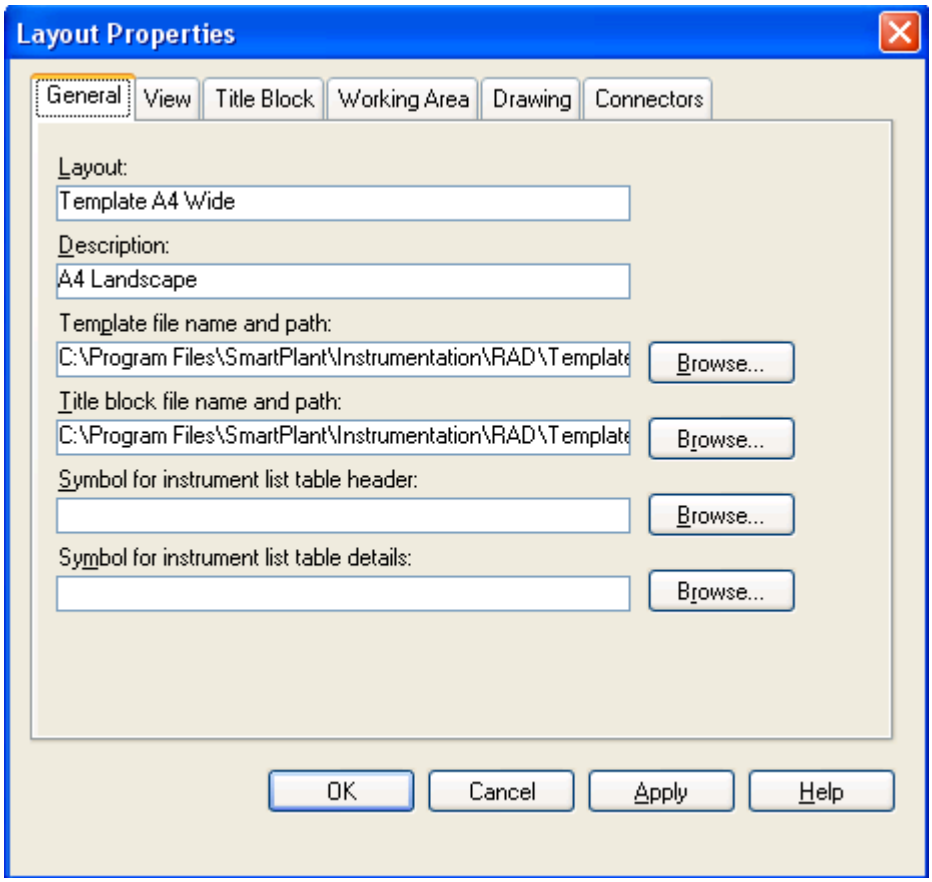
Task 4 — Changing the Enhanced Report Layout Properties

When you generate an Enhanced SmartLoop drawing, the software automatically calculates the optimal layout based on the number of points in the loop and the selected template size. If the loop changes, by the addition or deletion of elements, the page layout changes accordingly. In this task, you will configure some aspects of the appearance of the report manually by changing the values of some of the layout properties.

Define an Enhanced Report Layout and Assign the Layout to a Loop

1. In the **Domain Explorer**, select and then right-click loop **101-F -100**.
2. On the shortcut menu, click **Actions** > **Enhanced Report Layouts**.
3. On the **Enhanced Report Layouts** dialog box, click **New**.
4. On the **Layout Properties** dialog box, on the **General** tab, do the following:
 a. In the **Layout** box, type Template A4 Wide.
 b. In the Description box, type A4 Landscape.
 c. Beside **Template file name and path**, click **Browse**, navigate to the location <SmartPlant Instrumentation home folder>\RAD\Template\ and select the file Awide.sma.
 d. Beside **Title block file name and path**, click **Browse**, navigate to the location <SmartPlant Instrumentation home folder>\RAD\Template\Types\loop\ and select the file Loop_Wide.sym.

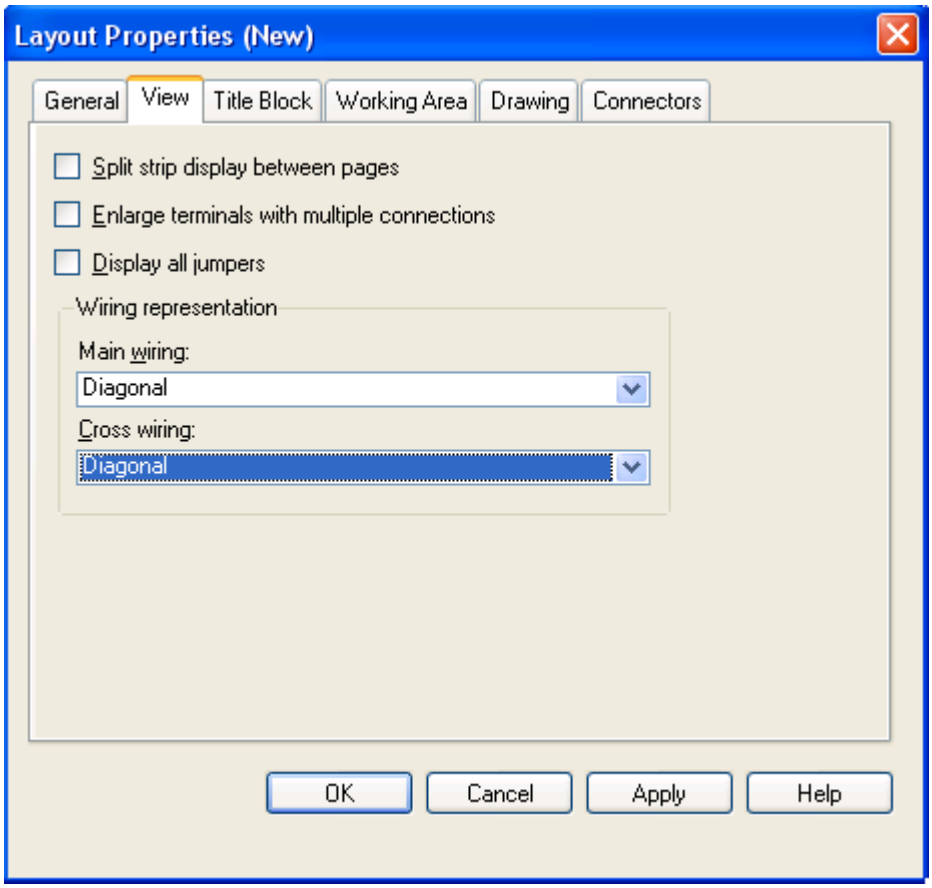
The dialog box should appear as shown:



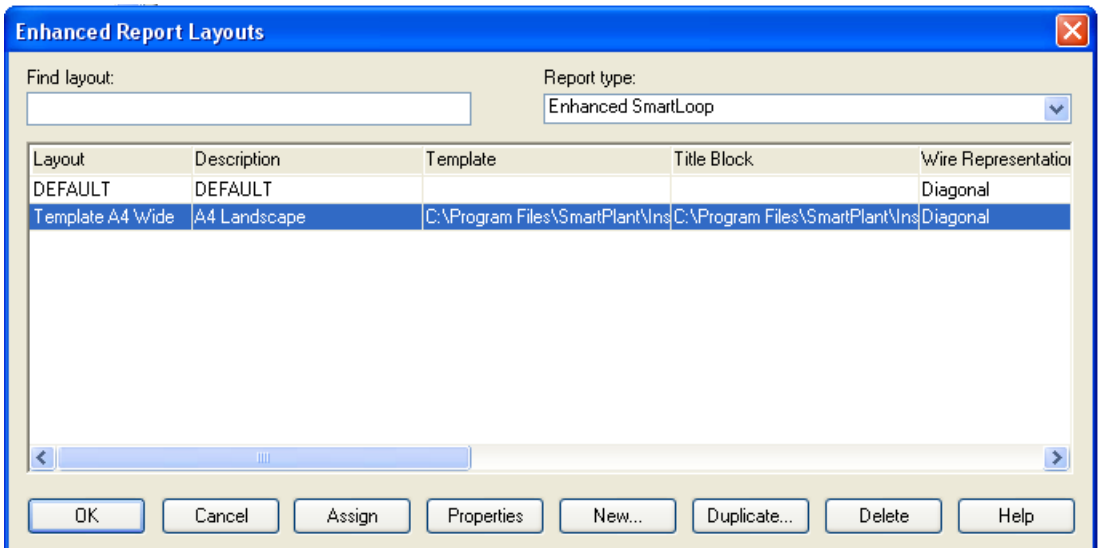
TIP The file paths may be different to the ones shown, depending on the folder where you installed SmartPlant Instrumentation.

5. On the **View** tab, under **Wiring representation**, select **Diagonal** from the **Main wiring** and **Cross wiring** lists.

The dialog box should appear as shown:



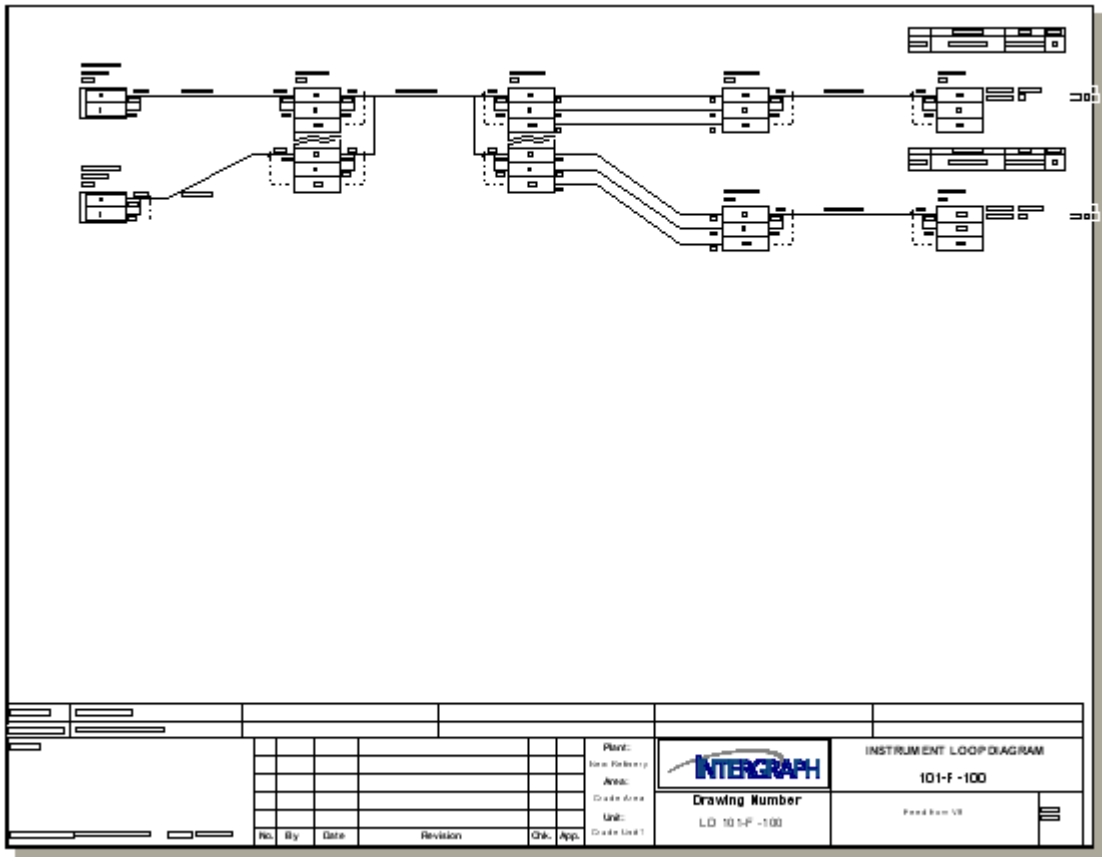
6. Click **OK** to close the dialog box and return to the **Enhanced Report Layouts** dialog box. The new layout appears as a highlighted row as shown:



7. Click **Assign** to assign the **Template A4 Wide** layout to loop **101-F -100**.
8. Click **OK** to close the dialog box.

9. When prompted to save data changes, click **Yes**.
10. Generate the Enhanced SmartLoop report for loop **101-F -100**.
11. On the **Main** toolbar, click the **Fit** command to display the entire report.

 The report should appear as shown:

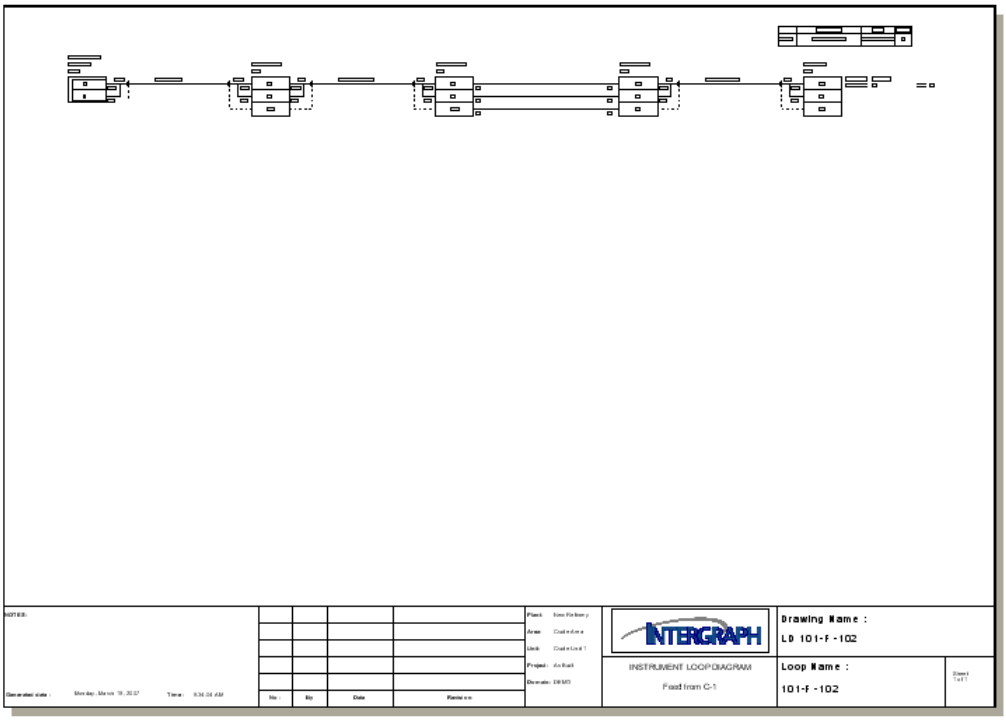


Assign an Enhanced Report Layout to Multiple Loops

Once you have defined a layout, you have the advantage of being able to assign it to a number of loops with similar drawing elements. In this way, you do not have to specify the layout for each loop individually.

1. In the **Domain Explorer**, select and then right-click loop **101-F -102**.
2. On the shortcut menu, click **Actions** > **Enhanced Report Layouts** and note that the **DEFAULT** layout is assigned to this loop.
3. Generate the report for loop **101-F -102**, making sure that you select **Enhanced Report** as the generation method.

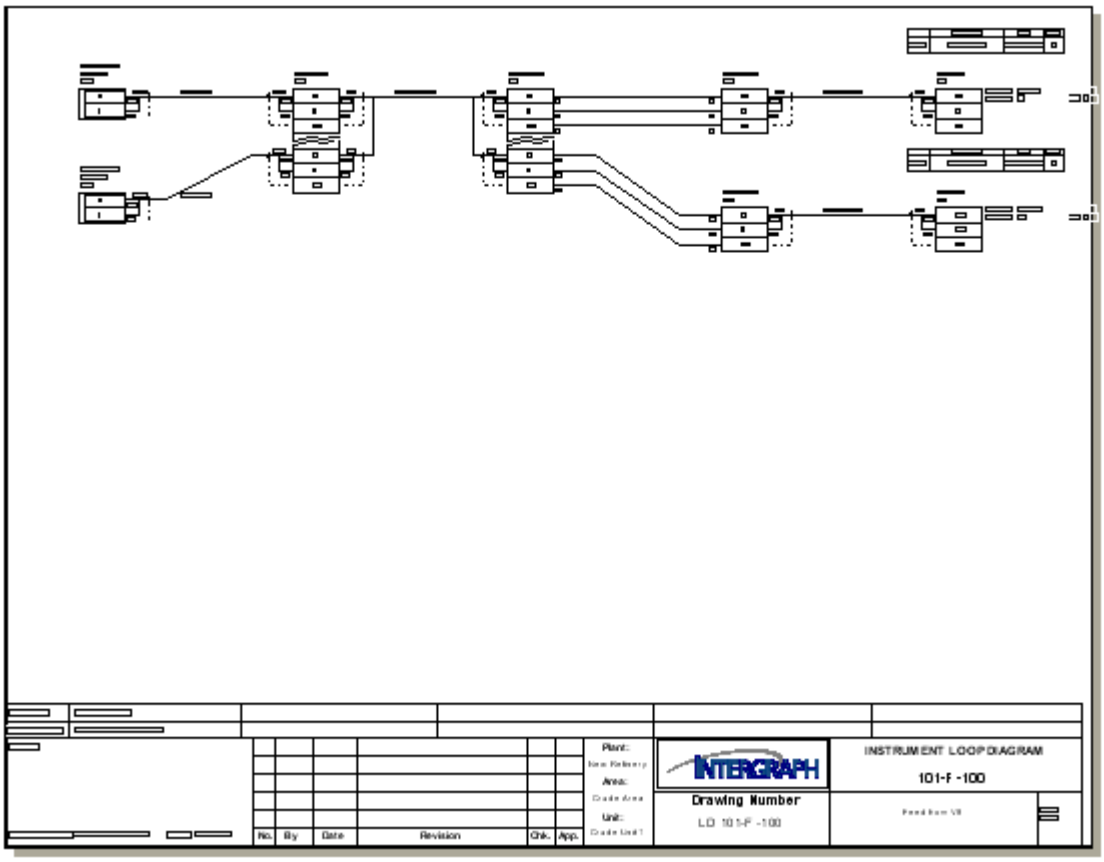
The report should appear as shown:



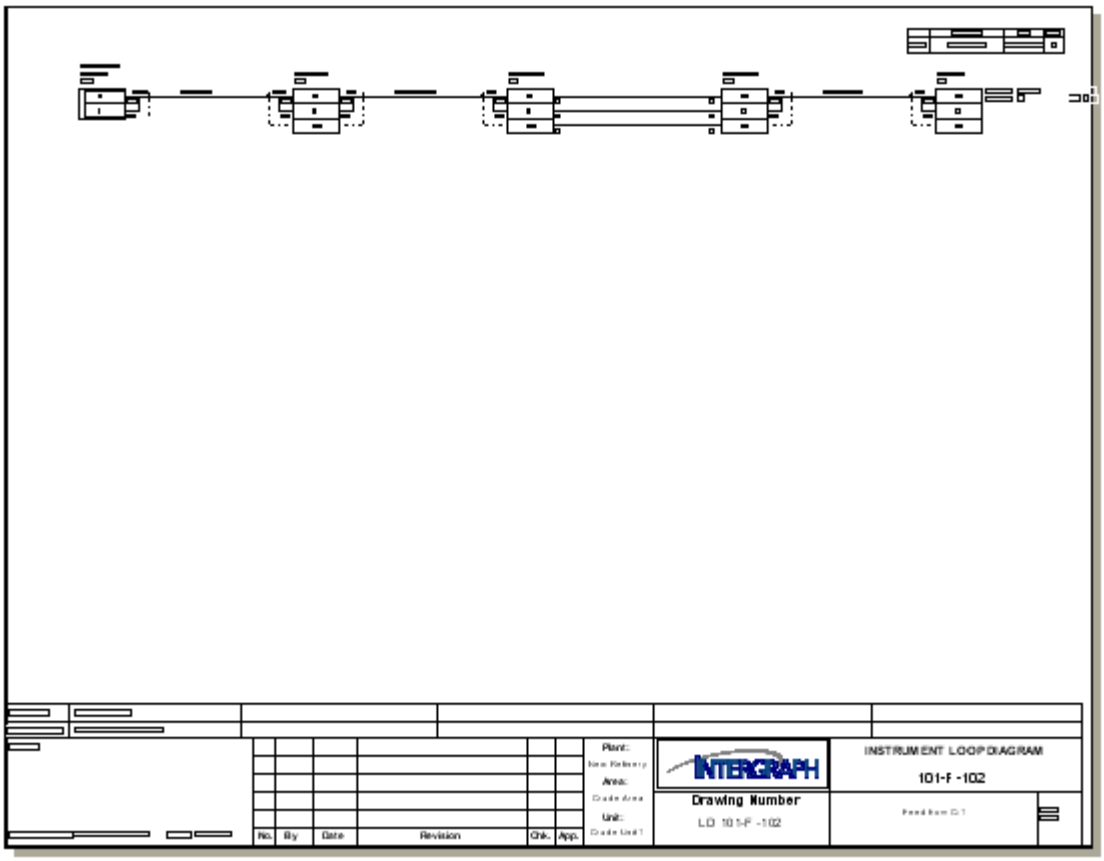
4. Return to the **Domain Explorer** and click the **Loops** folder.
5. In the **Items** pane, select loops **101-F -100** and **101-F -102**.
6. Right-click, and on the shortcut menu, click **Actions** > **Enhanced Report Layouts**.
7. In the message box notifying you that the highlighted layout is the one assigned to the first item in the selection, click **OK**.
8. On the **Enhanced Report Layouts** dialog box, select the layout **Template A4 Wide** and click **Assign**.
9. In the message box, click **OK**.
10. Click **OK** to close the **Enhanced Report Layouts** dialog box.

 TIP If you select loop **101-F -102** and open the **Enhanced Report Layouts** dialog box, you can see that the layout **Template A4 Wide** is now assigned to this loop.
11. In the **Domain Explorer**, in the **List** pane, select loops **101-F -100** and **101-F- 102**.
12. Right-click, and on the shortcut menu, click **Reports** > **Generate Loop Drawings** to generate the reports for both loops.
13. In the Enhanced Report Utility, view each report by selecting it from the **Window** menu.

The report for Loop 101-F- 100 should appear as shown:



The report for Loop 101-F- 102 should appear as shown:



Task 5 — Adding Macros or Macro Labels to an Enhanced SmartLoop Drawing

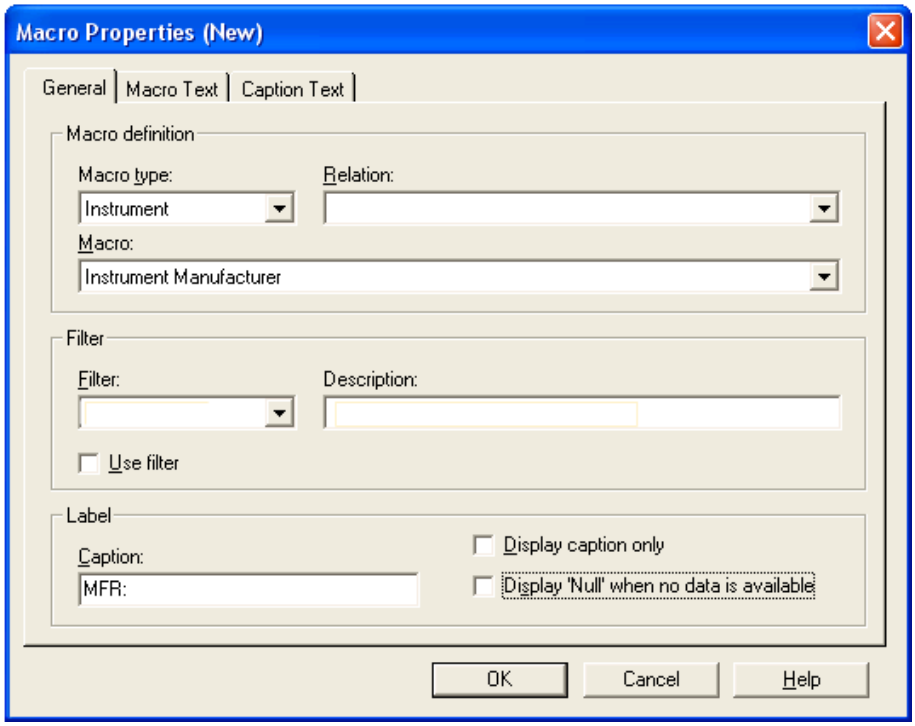
After generating a loop drawing, you may want to add macros to the drawing sheet or to specific items on the drawing sheet. In the following example you will add macros that display the manufacturers name and model associated with tag numbers.

Add a Macro or a Macro Label to a Tag Number

1. Generate the drawing for loop **101-F -100**.
2. In the Enhanced Report Utility, click tag **101-FT -100**.
3. Click **Actions** > **Macro** > **New**.

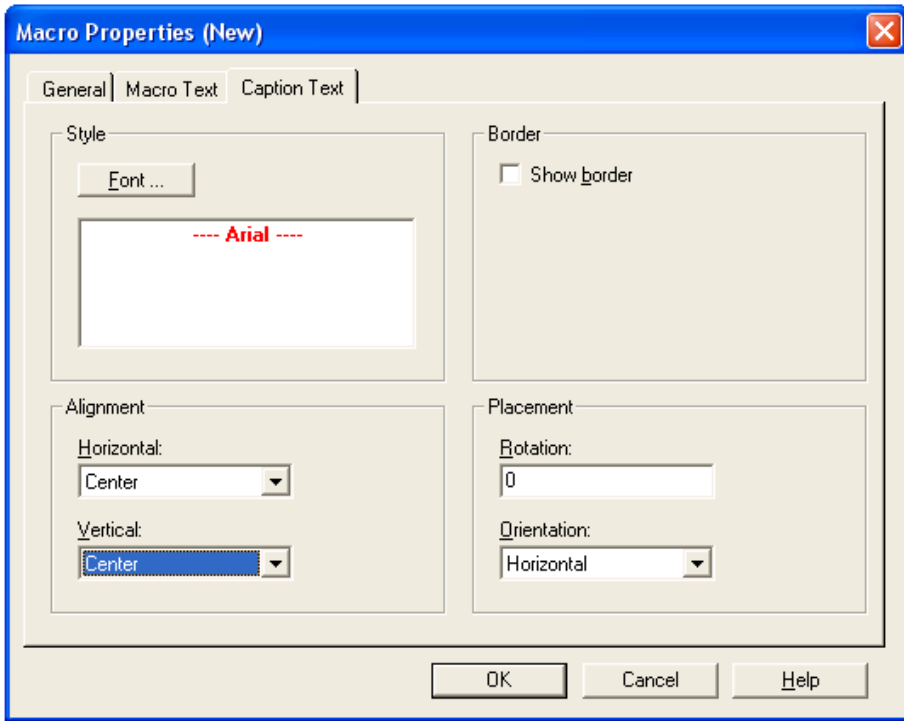
 TIP You can also access this command by clicking **New Macro** (⇨) on the **Actions** toolbar.
4. On the **Macro Properties** dialog box, on the **General** tab, do the following:
 a. From the **Macro** list, select **Instrument Manufacturer**.
 b. In the **Caption** box, type **MFR:**
 c. Clear the **Display 'Null' when no data is available** check box.

The dialog box should appear as shown:



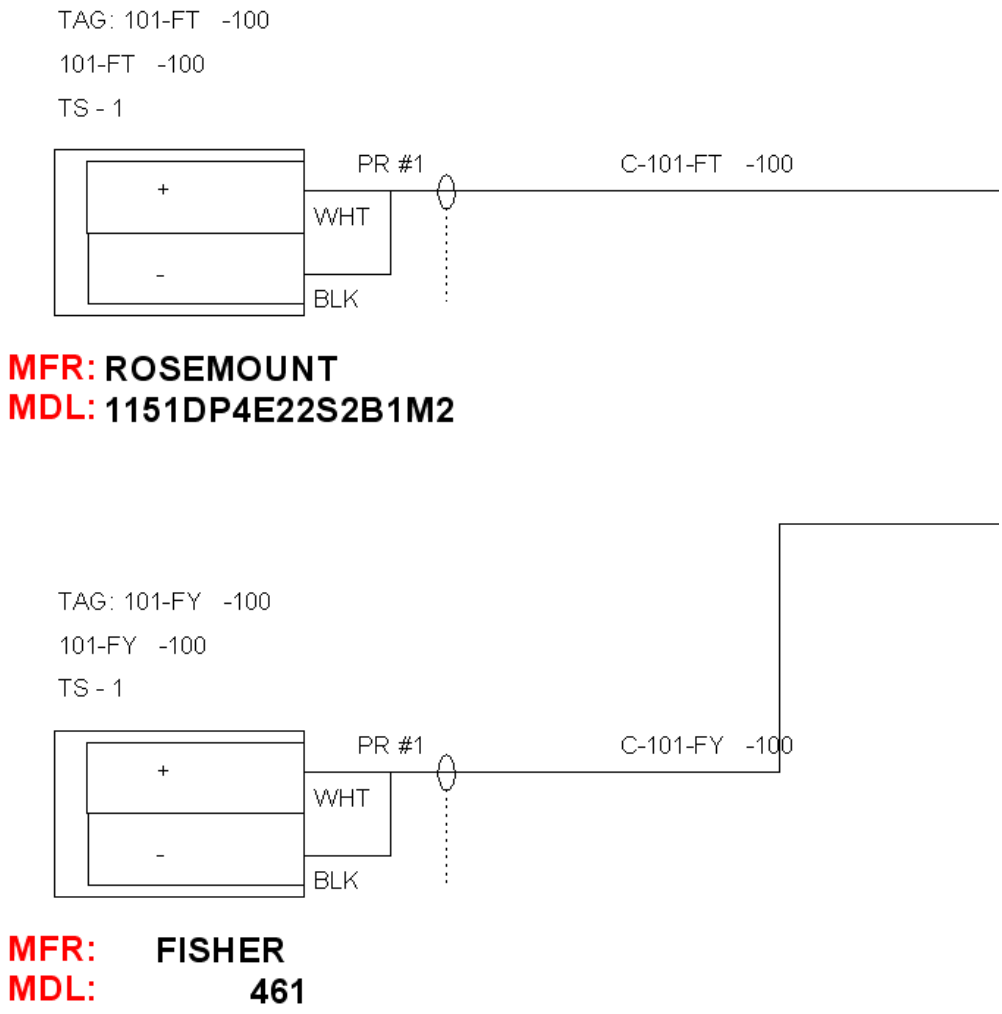
5. Click the **Macro Text** tab, and do the following:
 a. Click **Font** to open the **Font** dialog box.
 b. In the **Font style** field, select **Bold**.
 c. In the **Size** field, select **12**.
 d. In the **Color** field, select **Black**.
6. Click **OK** to return to the **Macro Properties** dialog box.
7. Under Alignment, for both Vertical and Horizontal fields, select Center.
8. Click the **Caption Text** tab, and repeat the steps for the **Macro Text** except set the color to **Red**.

The dialog box should appear as shown:



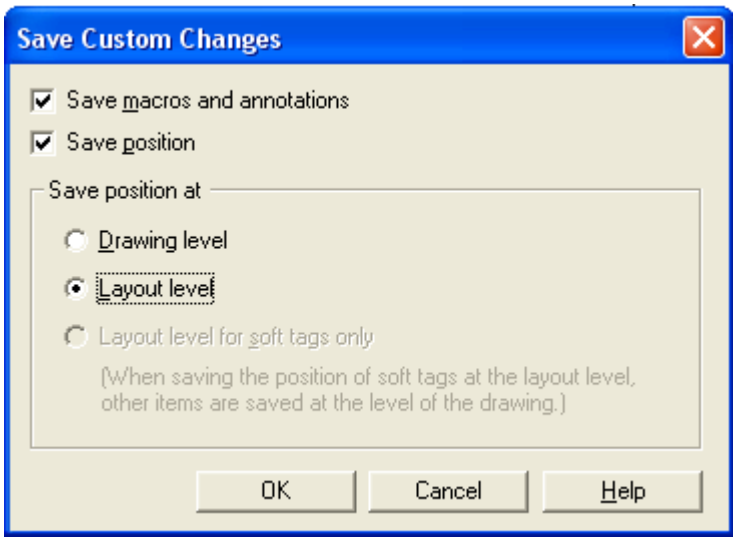
9. Click **OK** to return to the drawing sheet.
10. On the drawing sheet, click the cross-hair cursor where you want the macro to appear.
11. Click loop **101-F -100**.
12. Click **Actions** > **Macro** > **New**.
13. On the **Macro Properties** dialog box, on the **General** tab, do the following:
 a. From the **Macro** list, select **Instrument Model.**
 b. In the **Caption** box, type **MDL:**
 c. Clear the **Display 'Null' when no data is available** check box.
14. Define the same properties as for manufacturer on the **Macro Text** and **Caption Text** tabs.
15. Click **OK** to return to the drawing sheet.
16. On the drawing sheet, click below the previous macro.

The macros appear on the drawing as shown:



17. Click anywhere on the drawing sheet.
18. On the main toolbar, click **Save Custom Changes** ().

The **Save Custom Changes** dialog box opens as shown:



19. Click **OK**.

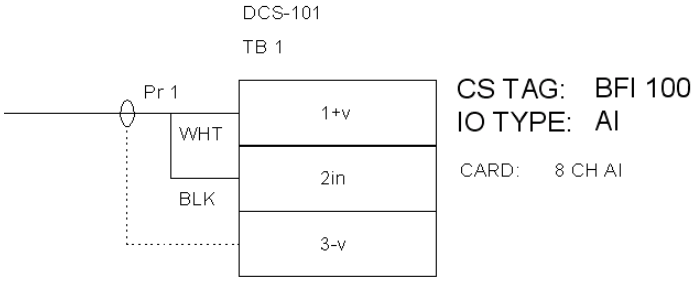
Attach a Macro or a Macro Label to Wiring Equipment

In the following example you will create macros associated with wiring equipment that show the name of the I/O card.

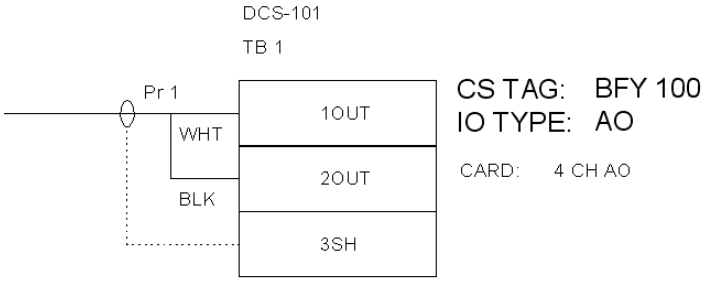
1. In the Enhanced Report Utility, generate the report for loop **101-F -100**.
2. Click **TB1** to select the terminal blocks.
3. Click **Actions** > **Macro** > **New**.

 TIP You can also access this command by clicking **New Macro** on the **Actions** toolbar.
4. On the **Macro Properties** dialog box, on the **General** tab, do the following:
 a. From the **Relation** list, select **Terminal Strip - Wiring Equipment**.
 b. From the **Macro** list, select **Wiring Equipment**.
 c. In the **Caption** box, type **Card:**
 d. Click **OK**.
5. On the drawing sheet, click where you want the macro to appear.

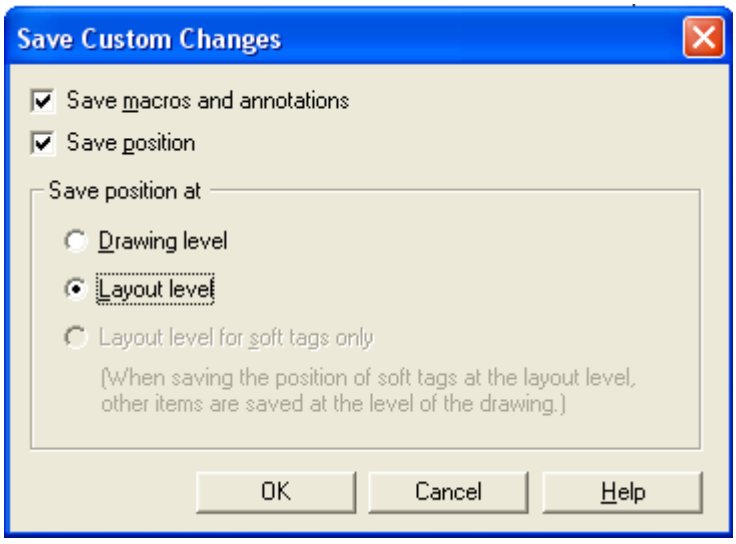
The macros appear, with the I/O Card name displayed on the drawing sheet as shown:



	CABINET	FILE	POS.
PRIM	DCS-101	File Number 1	2



6. On the main toolbar, click **Save Custom Changes** ().
 The **Save Custom Changes** dialog box opens as shown:



7. Click **OK**.

Define and Use Filters with a Macro

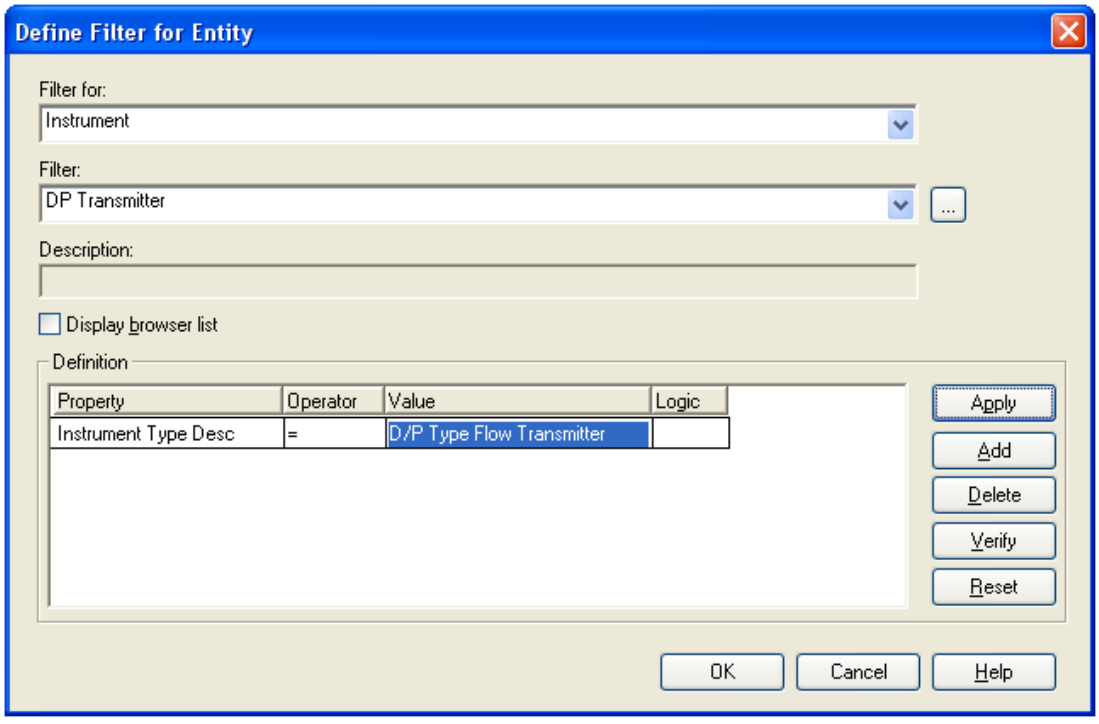
You can use filters with macros to show attributes for specific items. You can attach a filter when creating your macro, or define a filter and attach it to an existing macro. In this example you will define a filter that shows only the manufacturer and model of DP Transmitter.

1. Generate a drawing for loop **101-F -100**.
2. Click **Actions** > **Item** > **Define Filter**.

 TIP You can also access this command by clicking **Define Filter** () on the **Actions** toolbar.
3. Switch to SmartPlant Instrumentation to view the **Define Filter for Entity** dialog box, and do the following:
 a. From the **Filter for** list, select **Instrument**.
 b. From the **Filter** list, select **DP Transmitter**.

 TIP Click **Browse** ... to search for a filter or create a new filter name.
4. Click **Edit**, and then do the following:
 a. Click **Add**.
 b. From the **Property** list, select **Instrument Type Desc**.
 c. From the **Operator** list, select the operator equals (**=**).
 d. From the **Value** list, select **D/P Type Flow Transmitter**.

 The dialog box should appear as shown:

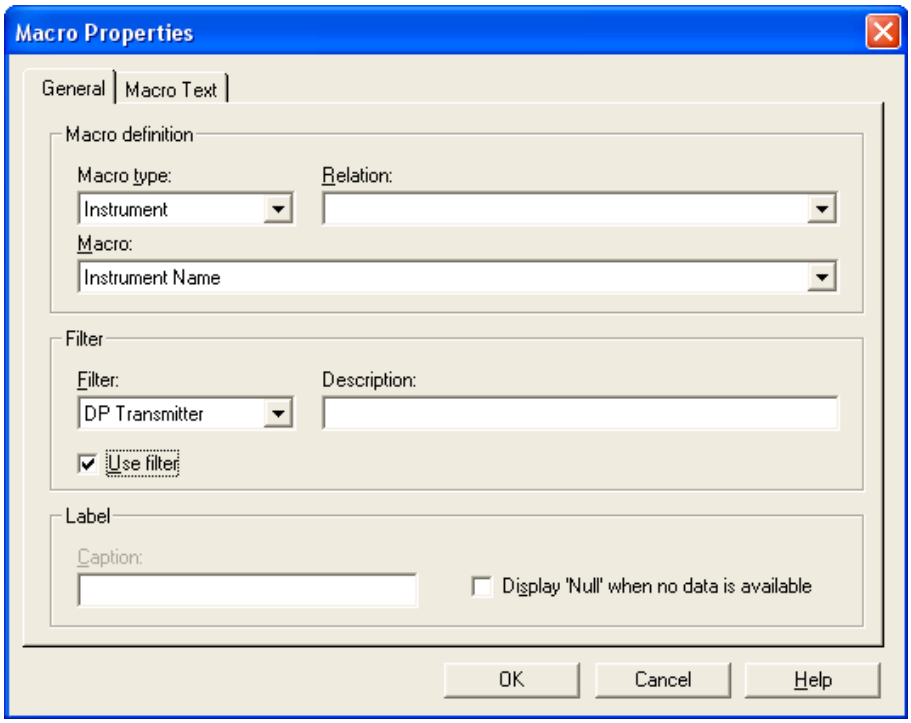


5. Click **Apply**, and then click **OK**.
6. On the generated drawing, click tag **101-FT -100**.
7. Do one of the following:
 - Click **Actions** > **Macro** > **Properties**.

- Click **Edit Macro Properties** () on the **Actions** toolbar.

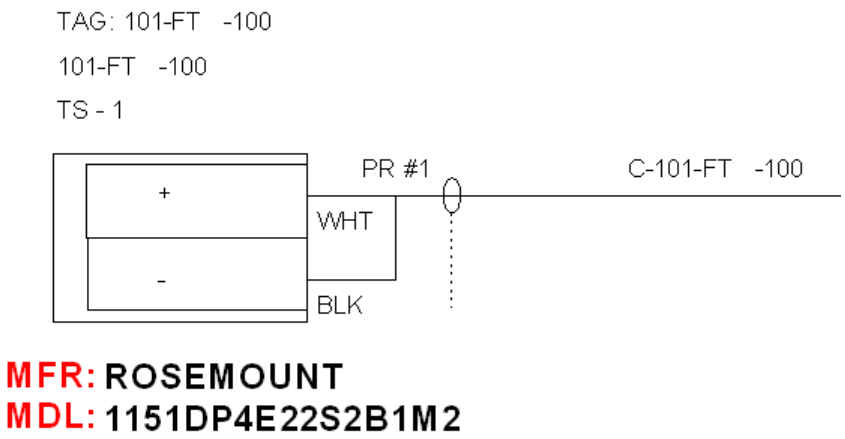
8. On the **Macro Properties** dialog box, from the **Filter** list, select **DP Transmitter**.
9. Select the **Use filter** check box.

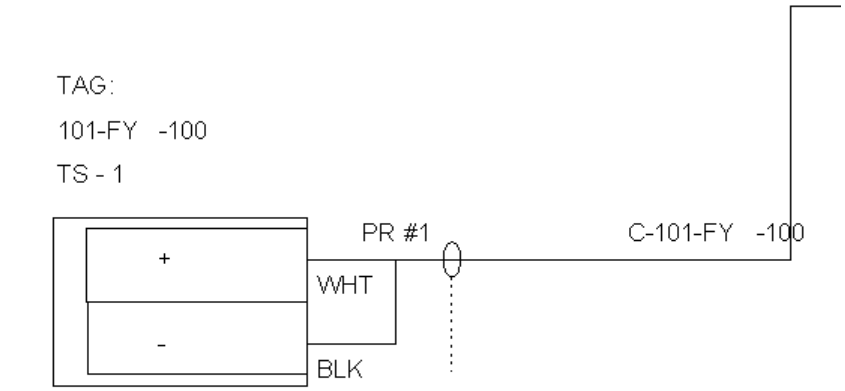
 The dialog box should appear as shown:



10. Click **OK**.
11. Click the macro label **MFR** for tag **101-FT -100**, and then do one of the following:
 - Click **Actions** > **Macro** > **Properties**.
 - Click  on the **Actions** toolbar.
12. On the **Macro Properties** dialog box, select the **Use filter** check box to activate the filter.
13. Click **OK**.
14. Repeat the procedure for the macro value **Rosemount**, and for the macro label **MDL** and its value for tag **101-FT -100**.

The filter hides the labels and values for those tags that are not DP Transmitters, as shown:





15. On the main toolbar, click **Save Custom Changes** ().
16. On the **Save Custom Changes** dialog box, click **OK**.

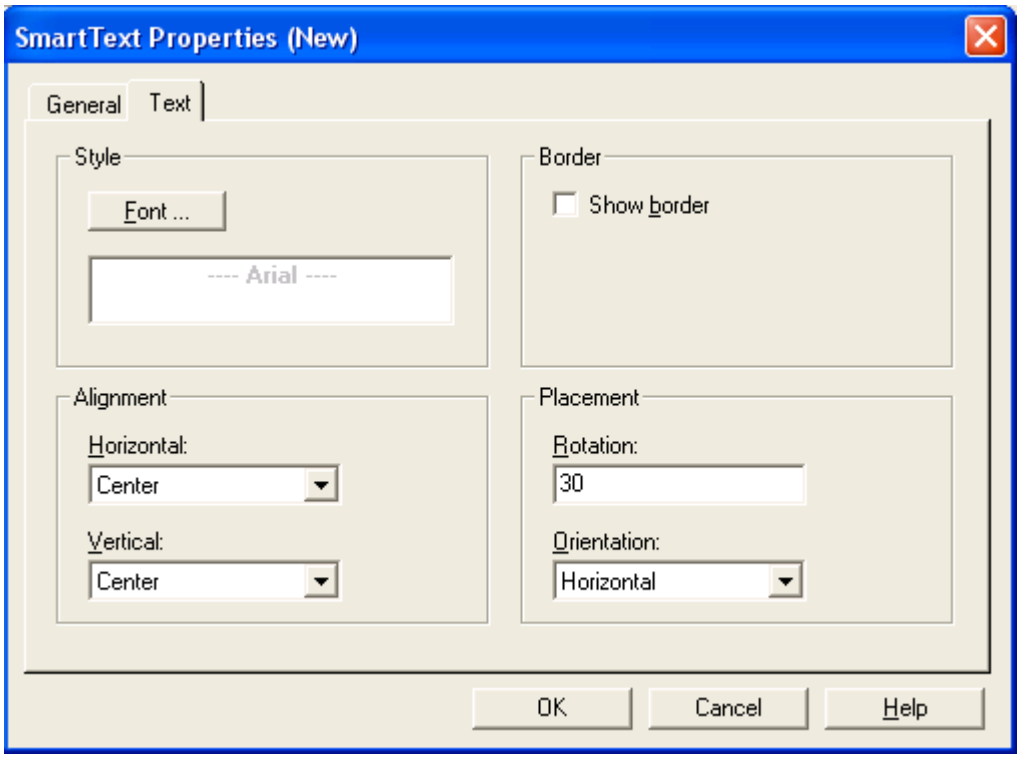
Task 6 — Adding Annotations to an Enhanced SmartLoop Report

During the process of creating control loops, you may want to revise loop parameters, correct faulty wiring, or indicate special operating conditions. You can annotate SmartLoop drawings by including a watermark and by adding redlining.

Include a Watermark

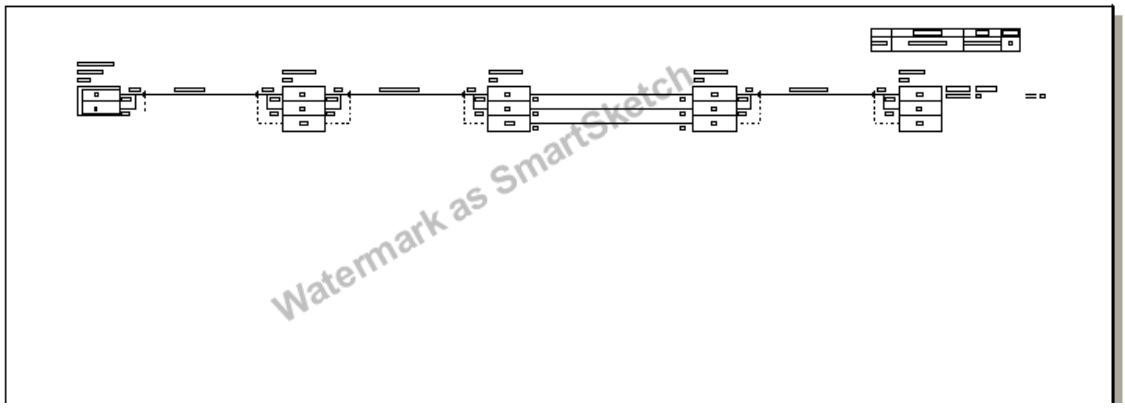
1. In the Enhanced Report Utility, generate the drawing for loop **101-F -102**, and then do one of the following:
 - Click **Actions** > **SmartText** > **New**.
 - Click **New SmartText** on the **Annotations** toolbar.
2. On the **SmartText Properties** dialog box, on the **General** tab, enter the following information:
 a. In the **Caption** field, type **Watermark as SmartText**.
 b. Under **Watermark**, select the **Set as watermark** check box.
3. Click the **Text** tab and do the following:
 a. Click **Font** to open the **Font** dialog box.
 b. In the **Font style** field, select **Bold**.
 c. In the **Size** field, select **72**.
 d. In the **Color** field, select **Silver**.
4. Click **OK** to return to the **SmartText Properties** dialog box.
5. Under **Alignment**, for both **Vertical** and **Horizontal** fields, select **Center**.
6. In the **Rotation** field, type **30** (degrees).

The dialog box tab should appear as shown:



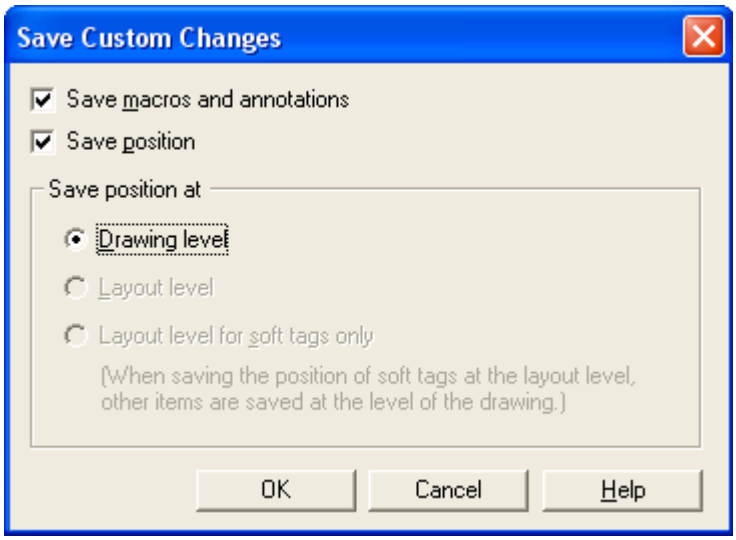
7. Click **OK** to return to the drawing sheet.
8. Place the cross-hair cursor at the center of the blank area of the sheet below the loop elements.

 The SmartLoop drawing is displayed with the watermark as shown:



9. Click the **Select Tool** to select the watermark and drag it on the drawing sheet as you desire.
10. On the main toolbar, click **Save Custom Changes** ().

The **Save Custom Changes** dialog box opens as shown:



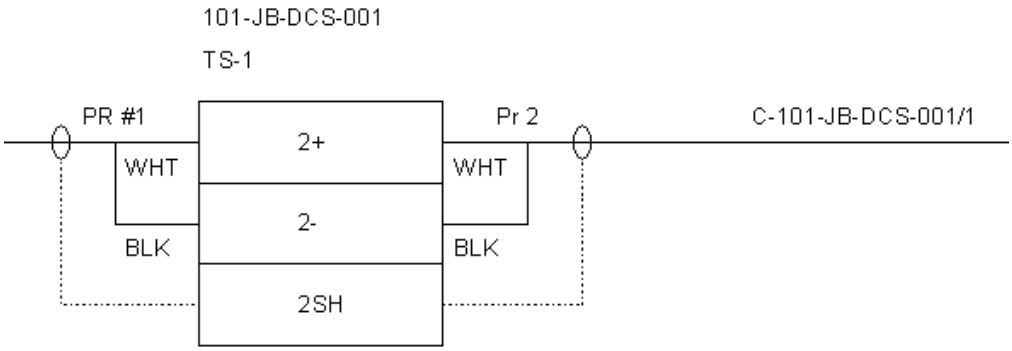
11. Click **OK**.

TIP Implementing the **Save Custom Changes** command ensures that whenever you generate the report for the loop, it appears with the watermark and any other annotations that you have added.

Add Redlining

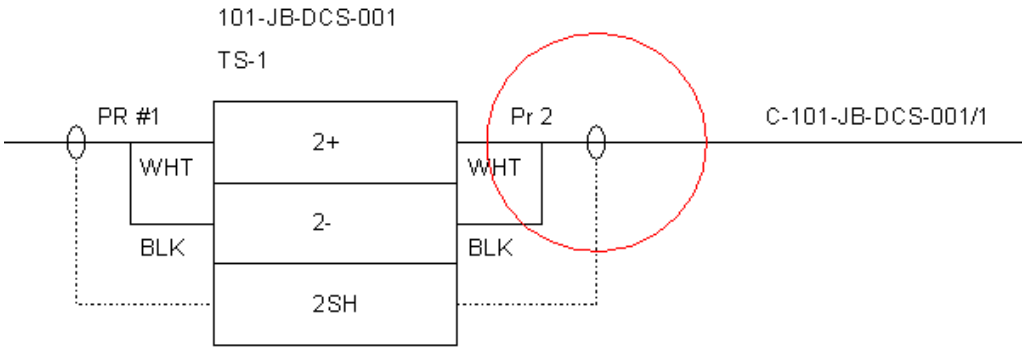
In this objective, you will mark the wires on the loop and add a note to show that the wire colors should be switched.

1. In the Enhanced Report Utility, generate the drawing for loop **101-F -102**.
2. On the main toolbar, click the **Zoom Area** command and drag the zoom area over the field device and adjacent connector on the drawing to display more details.



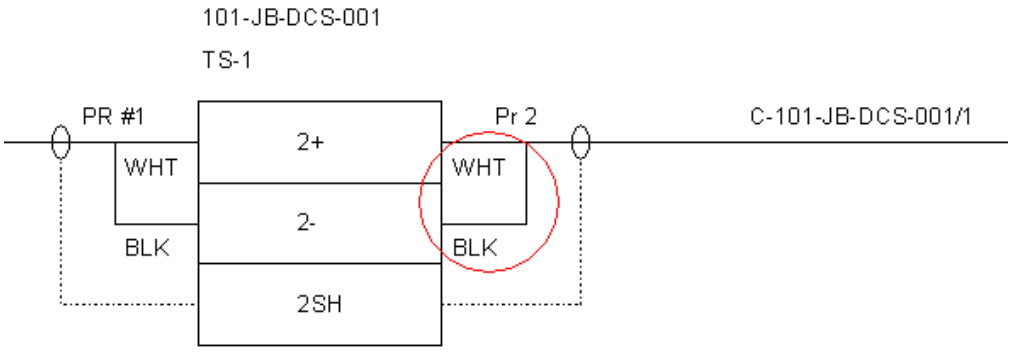
3. On the **Draw** toolbar, click , move the cross-hair cursor in the drawing area to the position where you want to place the center of the circle, and click the cursor once at that position.

4. Release the mouse button and drag the mouse until the circle is of the desired size, then click to place the circle.



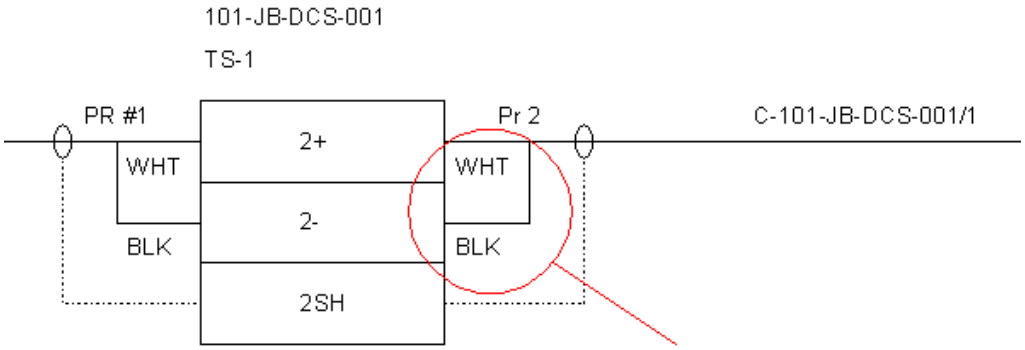
TIP You can change the line color, width, and style of the active object with the Line ribbon that appears when drawing an object.

5. Position and size the circle as shown as follows:
 a. Click and then click anywhere on the circumference of the circle.
 b. Select the handle at the center of the circle and drag to move the circle.
 c. Select the handle at the edge of the circle and drag to resize the circle.
 d. By following the above steps, try to position and size the circle approximately as shown.

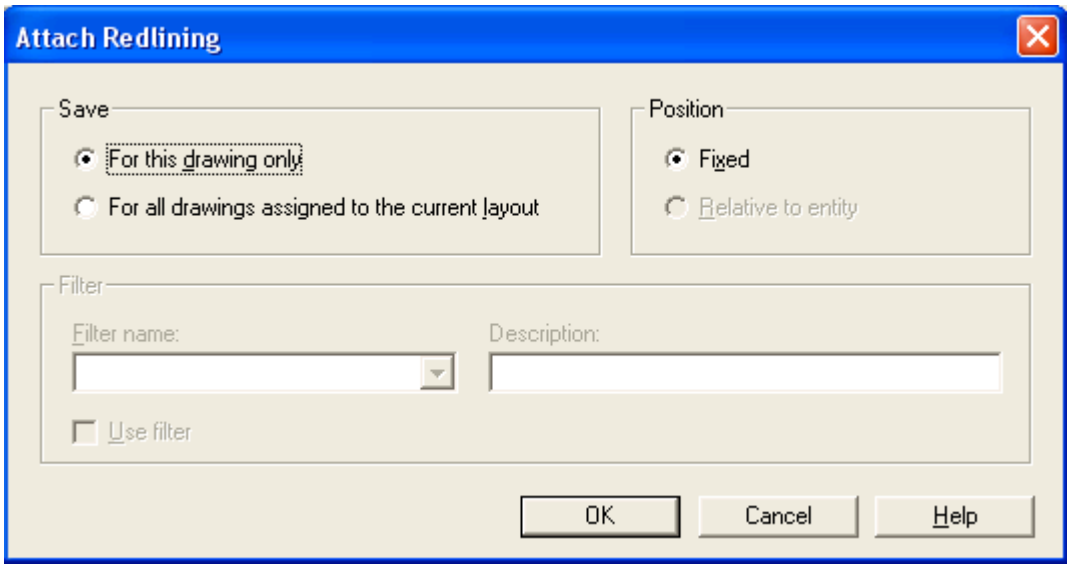


6. Draw a line next to the circle as follows:
 a. Click to start the drawing line mode and move the cursor to the drawing area.
 b. Click once below and to the right of the circle about two inches away to define the start of the line.
 c. Click again at the lower edge of the circle to define the end of the line.
 d. Click to exit the drawing line mode.

The line should appear approximately as shown.

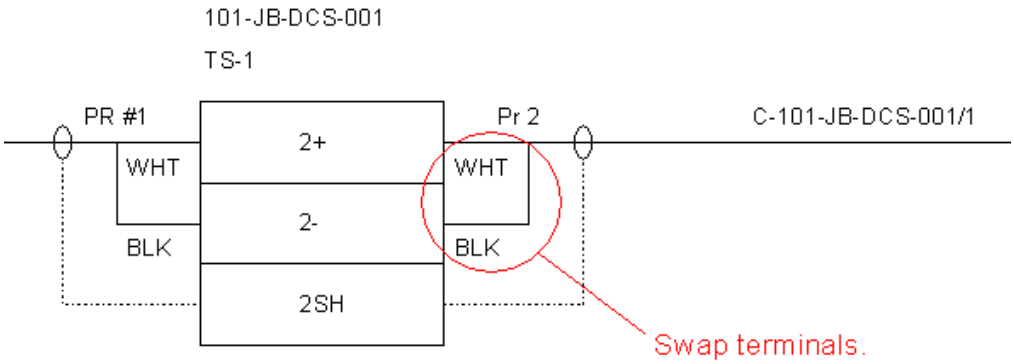


7. Click the circle to select it.
8. On the **Annotations** toolbar, click **Attach Redlining** ().
9. On the **Attach Redlining** dialog box, select **For this drawing only**.



10. Now do the same for the line that you drew.
11. Finally, add some text beside the line as follows:
 a. On the **Annotations** toolbar, click **New SmartText** .
 b. On the **SmartText Properties** dialog box, in the **Caption** field, type **Swap terminals**.
 c. Click the **Text** tab and click **Font** to open the **Font** dialog box.
 d. In the **Size** field, select **10**.
 e. In the **Color** field, select **Red**.
 f. Click **OK** to return to the **SmartText Properties** dialog box.
 g. Under **Alignment**, for both **Vertical** and **Horizontal** fields, select **Center**.
 h. Accept the remaining values and click **OK** to close the **SmartText Properties** dialog box and return to the drawing sheet.

i. Click the cross-hair cursor  at the lower right end of the line you drew. Drag the text so that it appears as shown.

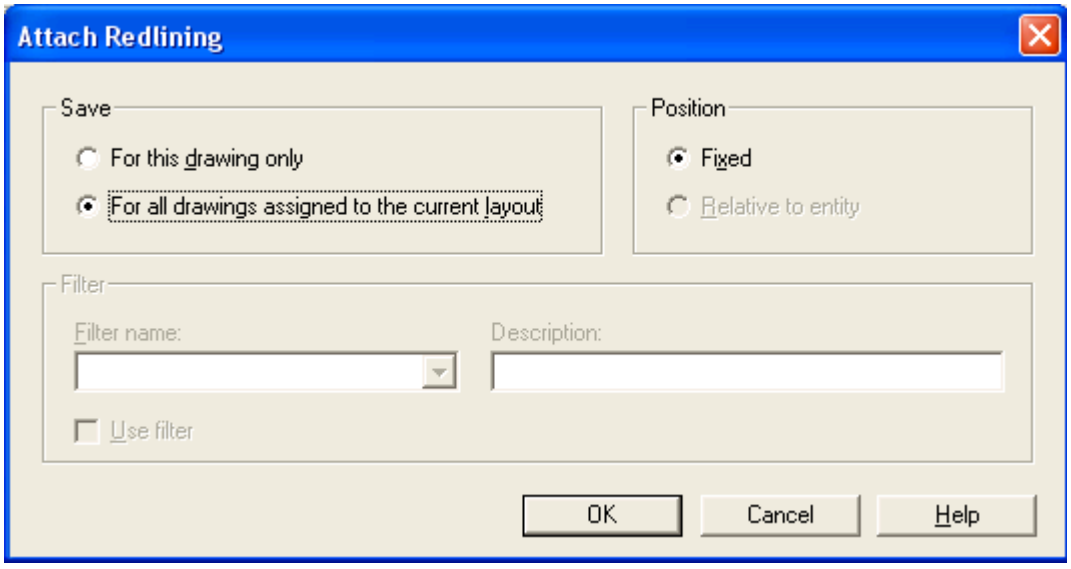


12. Click **File** > **Save Custom Changes**.
13. On the **Save Custom Changes** dialog box, in the **Save position at** box, select **Drawing level** and click **OK**.

Attach Redlining Items to a Layout

A powerful feature of the Enhanced Report Utility is the ability to attach redlining items per layout so that the software displays those items for all loops to which that layout is assigned.

1. Click the circle to select it.
2. On the **Actions** toolbar, click **Attach Redlining** ().
3. On the **Attach Redlining** dialog box, click **For all drawings assigned to the current layout**.



4. Repeat the above steps for the straight line.
5. On the **Main** toolbar, click **Save Custom Changes** ().
6. On the **Save Custom Changes** dialog box, in the **Save position at** box, select **Layout level** and click **OK**.

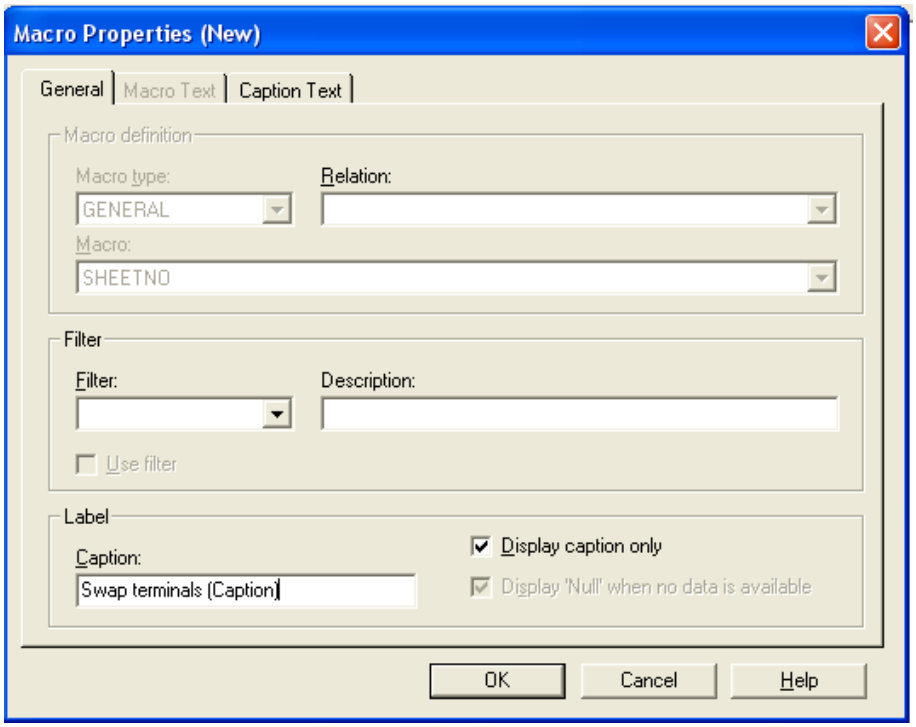
7. Close the Enhanced Report Utility and regenerate drawings for loops **101-F -100** and **101-F -102**.

NOTE The redlining items (circle and line) should appear in both reports; however, the software saves SmartText per loop only. To save text at the layout level, you must add it as a macro caption.

Add a Macro Caption

When you add SmartText to a drawing sheet and save to the database, the software displays the text every time you generate an Enhanced SmartLoop report for the specified loop. In this exercise, you will add text as a macro caption so that the text appears in all loop drawings that belong to the same enhanced report layout.

1. Display the drawing for loop **101-F -102**.
2. Click a blank area of the drawing sheet, and on the **Annotations** toolbar, click **New Macro** .
3. On the **Macro Properties** dialog box, in the **Label** box, under **Caption**, type **Swap terminals (Caption)**.
4. Select the **Display caption only** check box.



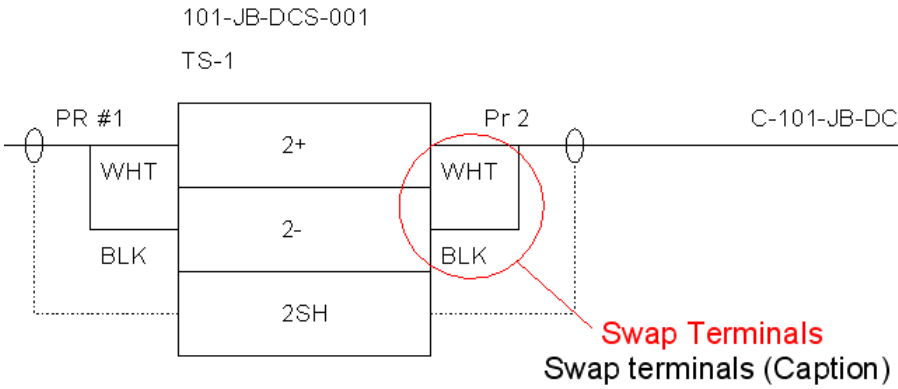
5. Click the **Caption Text** tab and do the following to set the values for the text formatting options:
 a. Click **Font** to open the **Font** dialog box.
 b. In the **Size** field, select **10**.
 c. Click **OK** to return to the **Macro Properties** dialog box.
 d. Under **Alignment**, for both **Vertical** and **Horizontal** fields, select **Center**.
 e. Accept the remaining values and click **OK** to close the **Macro Properties** dialog box and return to the drawing sheet.

f. Click the cross-hair cursor on the drawing sheet at the position where you want to insert the text.

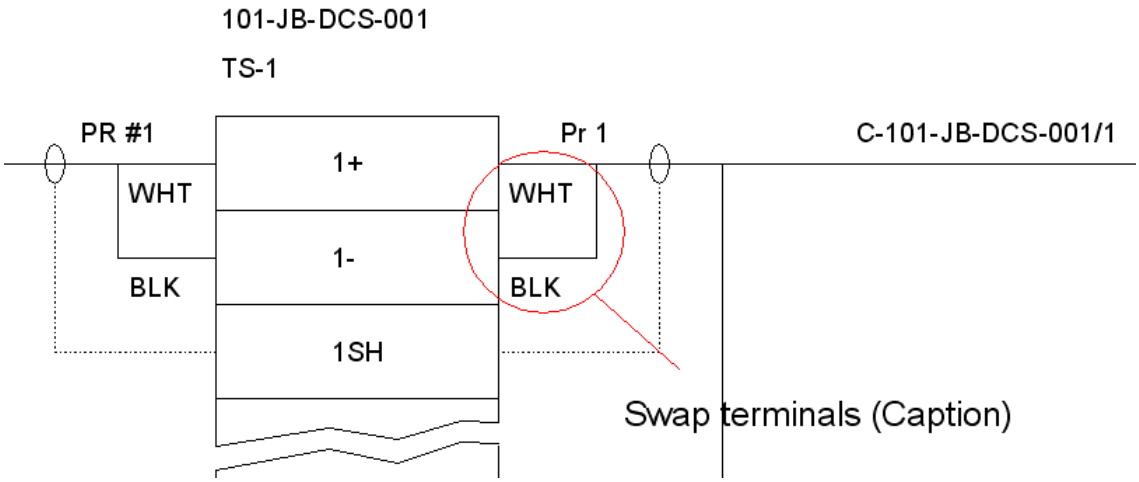
g. Drag this text immediately below the existing **Swap terminals** label that you added as SmartText.

6. On the main toolbar, click **Save Custom Changes** ().
7. On the **Save Custom Changes** dialog box, in the **Save position at** box, select **Layout level** and click **OK**.
8. Close the Enhanced Report Utility and regenerate drawings for loops **101-F -100** and **101-F -102**.

 The redlining and text should now appear in the drawing for loop 101-F -102 drawing as shown:



 The redlining and text should now appear in the drawing for loop 101-F -100 as shown:



NOTE SmartText entries are saved only on the drawing level and not on the layout level.

This completes the set of objectives for Enhanced SmartLoop drawings.

Generating CAD Loop Drawings

This section describes how to generate loop drawings using an external CAD engine such as SmartSketch, AutoCAD, or MicroStation.

The CAD session is based on SmartSketch. The procedures for AutoCAD and MicroStation are very similar.

Conventional loop drawings are generated from drawing blocks on which SmartPlant Instrumentation data is superimposed. The great advantage of this method is that from a small number of blocks, different combinations of instrument, wiring, and control panel arrangements can be obtained, and many hundreds, or thousands, of loop drawings, each with unique data, may be generated from a single pattern. Before starting the module, make sure that the following files are located in the path <SmartPlant Instrumentation home folder>\CAD\Blocks.

Block	SmartSketch File	AutoCAD File	MicroStation File
Logo	logo_dem.sym	logo_dem.dwg	logoig.cel
Border	bord_def.sym	bord_def.dwg	border.cel
Orifice Plate	fe_orif.sym	fe_orif.dwg	fe_ori.cel
Control Valve	cv.sym	cv.dwg	cv.cel
Analog Input	ai.sym	ai.dwg	ai.cel
Analog Output	ao.sym	ao.dwg	ao.cel

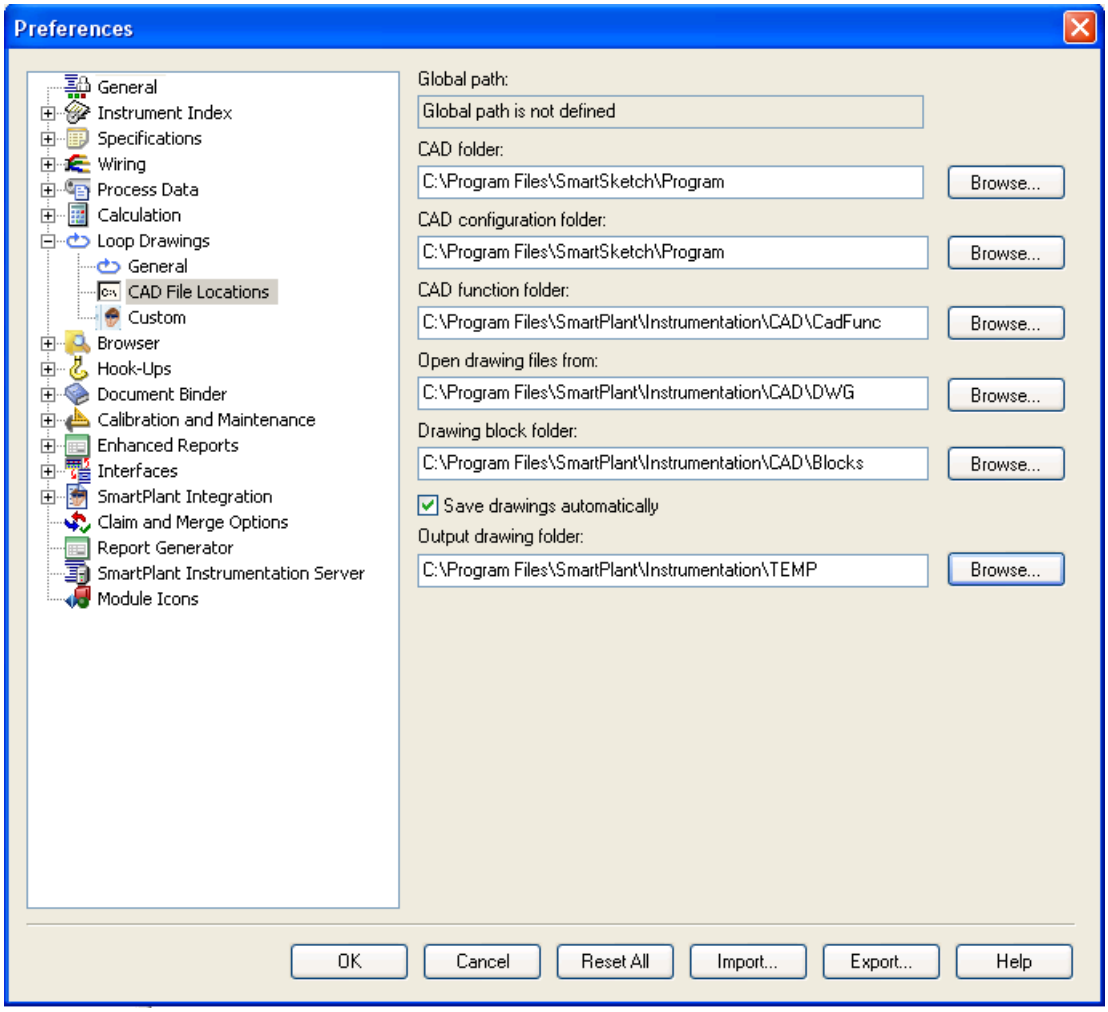
NOTE The purpose of this tutorial is not to instruct you how to define blocks or cells but rather to demonstrate how to generate loop drawings using a CAD application. At this stage, we have provided the necessary blocks to use in this Tutorial.

Task 1 — Setting Preferences

Set Preferences for your CAD Application

1. Click **File** > **Preferences**.
2. In the tree view, select **Loop Drawings** > **General**.
3. Under **Default generation method**, select **SmartSketch**.
4. Set other options on this page as appropriate for the specified application.
5. In the tree view, select **Loop Drawings** > **CAD File Locations**.

6. Set the folder locations for each group of files as appropriate for the specified CAD application as shown in the following example.



IMPORTANT You must specify the folder locations for the software be able to create drawing blocks and generate loop drawings using the CAD application that you selected.

7. On the **Preferences** > **General** page:
 a. Ensure that you have defined a value for the Temporary folder path.
 b. Tick the **Use macro symbol (&) to retrieve data** box.
8. Click **OK**.

NOTE If the Loop Drawings module is already open when you set the preferences, you must close the module and then re-open it for the settings to take effect.

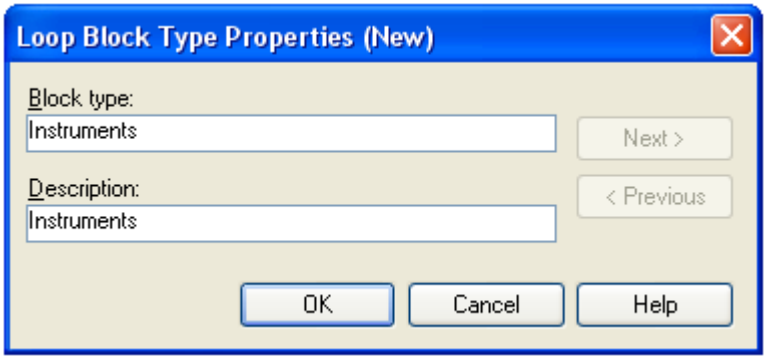
Task 2 — Defining Instrument Blocks

An instrument block is a drawing b lock you associate with instrument tags. Before you can assign blocks to instruments, you must first create an instrument block type. Classification of drawing blocks according to block type is useful for filtering the blocks. It is recommended that each block type contain drawing blocks that serve the same general function. This way, it is easy to find a particular drawing block when you want to use it.

You create instrument block type and define instrument blocks in the **Drawing Block Types** folder of the **Domain Explorer**. Blocks that you add in the **Drawing Block Types** folder become available for association with instruments.

Define an Instrument Block Type

1. In the **Domain Explorer**, right-click the **Drawing Block Types** folder.
2. On the shortcut menu, click **New** > **Instrument Block Type**.
3. On the **Loop Block Type Properties** dialog box, type data as shown.



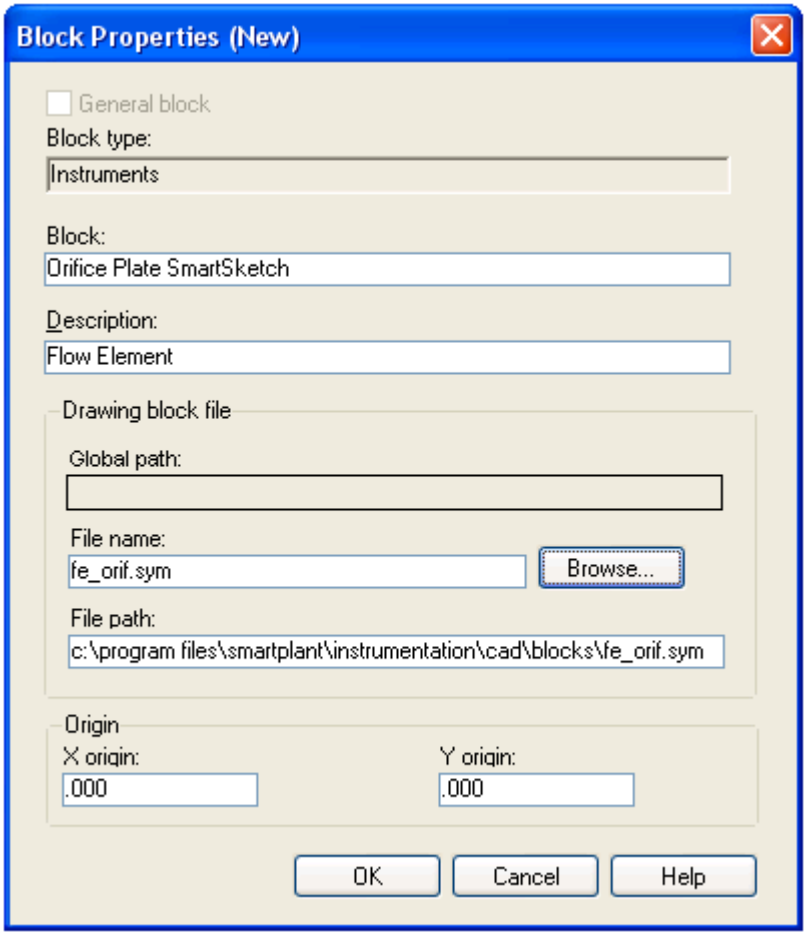
4. Click **OK**.

Define Instrument Blocks

Within each block type, you now need to define the drawing blocks and select for each block the file that contains the required drawing.

1. In the **Domain Explorer**, navigate to the **Instruments** block type, in the **Drawing Block Types** folder.
2. On the **Instruments** block type, right-click and from the shortcut menu select **New** > **Block**.
3. On the **Block Properties (New)** dialog box, enter data as shown.

TIP If you need to search for the appropriate file, click **Browse**.



4. Click **OK** to display the new block under **Instruments** block type.
5. Repeat steps 3 – 4 to add the following blocks to the **Instruments** block type:

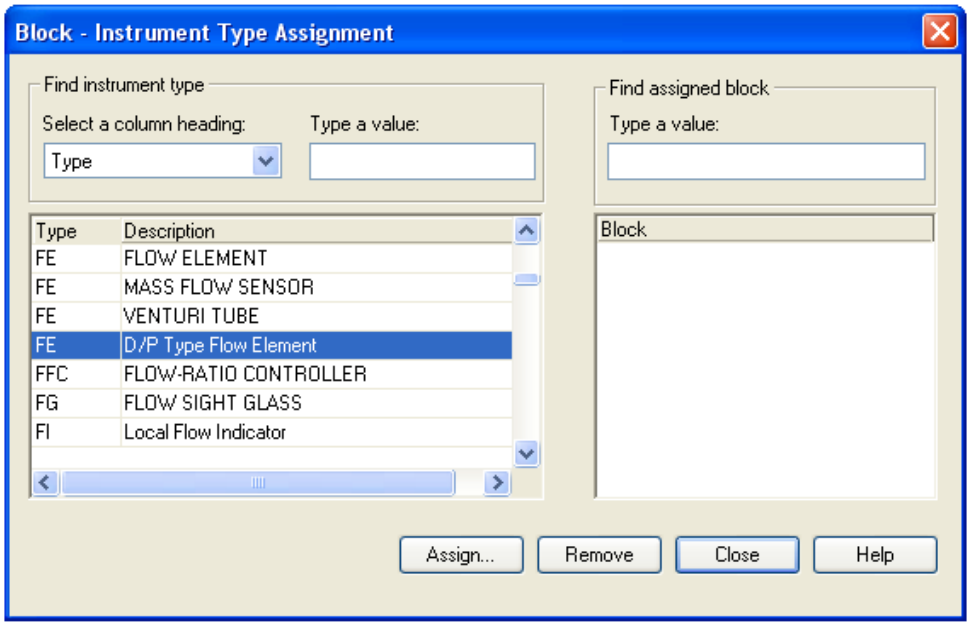
SmartSketch File	Block Name	Description
cv.sym	Control Valve SmartSketch	CONTROL VALVE
ai.sym	Analog Input SmartSketch	INPUT LOOP
ao.sym	Analog Output SmartSketch	OUTPUT LOOP

Task 3 — Assigning Blocks to Instrument Types

When you assign a block to a specific instrument type, the software associates this block automatically with any instrument tags of that instrument type. Such blocks appear green in the **Domain Explorer** when assigned to instruments. Only one block can be assigned to each instrument type.

Assign Blocks to Instrument Types

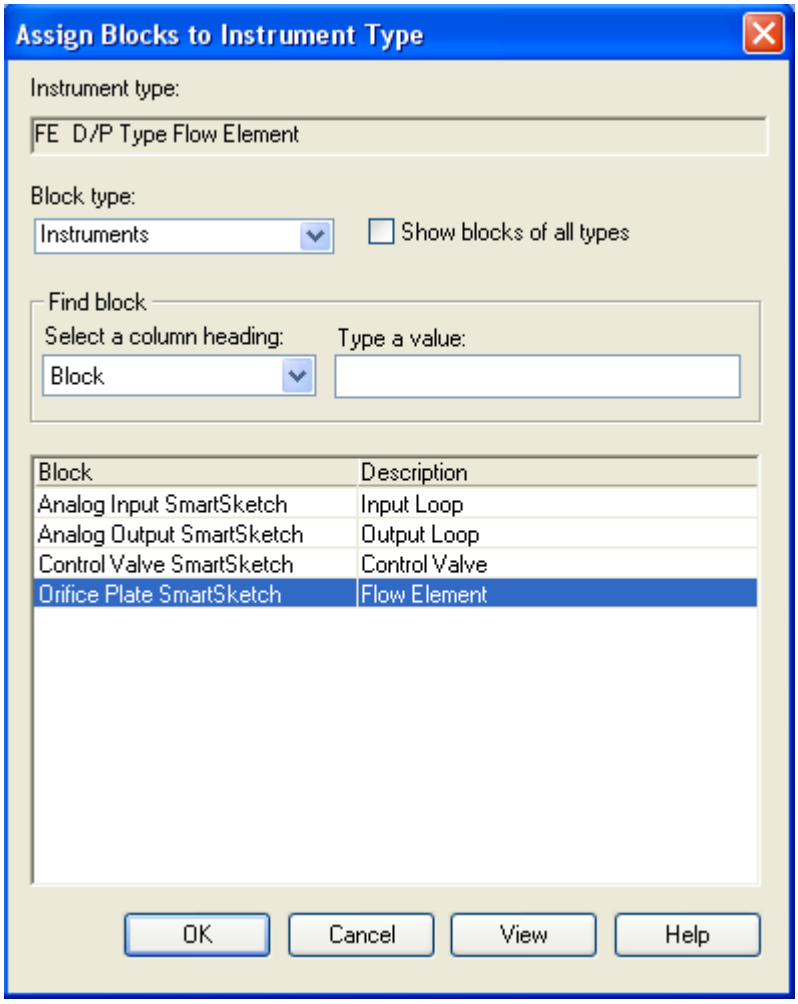
1. Open the Loop Drawing module by doing one of the following:
 a. On the main toolbar, click .
 b. On the main menu bar, click **Modules** > **Loop Drawings**.
2. Do one of the following:
 - On the **Loop Drawings Module** window toolbar, click .
 - On the **Loop Drawings Module** window menu bar, click **Actions** > **Block - Instrument Type Assignment**.
3. On the **Block – Instrument Type Assignment** dialog box, scroll down the list of instrument types and select **FE – D/P TYPE FLOW ELEMENT** as shown.



TIP This selection will only create associations with those instruments of type **FE** which are described as **D/P Type Flow Element**, and not with other **FE** descriptions.

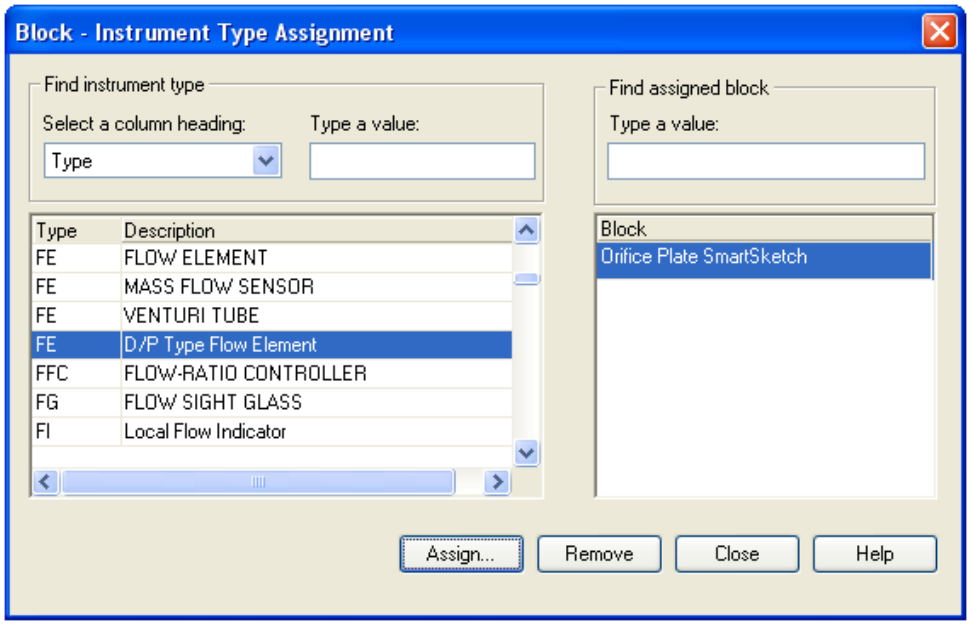
4. Click **Assign**.

5. On the **Assign Block To Instrument Type** dialog box, from the **Block type** list, select **Instruments**, and then in the data window select **Orifice Plate SmartSketch**.



6. Click **OK** to assign this block to the **FE D/P Type Flow Element** instrument type.

The **Block – Instrument Type Assignment** dialog box should appear as shown:



7. Make other block – instrument type assignments as shown in the following table.

Instrument Type	Description	Block Name
FT	D/P Type Flow Transmitter	Analog Input SmartSketch
FV	Control Valve	Control Valve SmartSketch
FY	I/P Transducer	Analog Output SmartSketch

8. When done, click **Close**.

Task 4 — Defining Border and Logo Blocks

Border and logo blocks are general blocks that appears in every CAD loop drawing that you generate. You do not associate these blocks with specific instruments. After adding your border and logo blocks in the **Logo** and **Border** block types in the **Domain Explorer**, you then need to set one logo and one border block as the default blocks for your CAD drawings. Using the following procedure, you can define your own border and logo using SmartSketch blocks.

Define New Border and Logo Blocks

1. In the **Domain Explorer**, in the **Drawing Block Types** folder, select the **Border** folder.
2. Right-click the **Border** folder, and from the shortcut menu click **New** > **Block**.
3. Define a new border block as follows:
 a. On the **Block Properties** dialog box, name the block **Demo Border SmartSketch**.
 b. Beside **File name**, click **Browse** and navigate to the file **bord_def.sym**.
 c. Click **OK** to close the **Block Properties** dialog box.
4. In the **Drawing Block Types** folder, select the **Logo** folder.
5. Right-click the **Logo** folder, and from the shortcut menu click **New** > **Block**.
6. Define a new logo block as follows:
 a. On the **Block Properties** dialog box, name the block **Demo Logo SmartSketch**.
 b. Beside **File name**, click **Browse** and navigate to the file **logo_dem.sym**.
 c. Click **OK** to close the **Block Properties** dialog box.
7. To view your new blocks, right-click the block you want to view, and from the shortcut menu, click **Reports** > **View Block in CAD Application**.

Set the Border and Logo Blocks as Default

1. Click **Actions** > **Default General Blocks**.
2. From the **Border block** list, select **Demo Border SmartSketch**.
3. From the **Logo block** list, select **Demo Logo SmartSketch**.
4. Click **OK**.

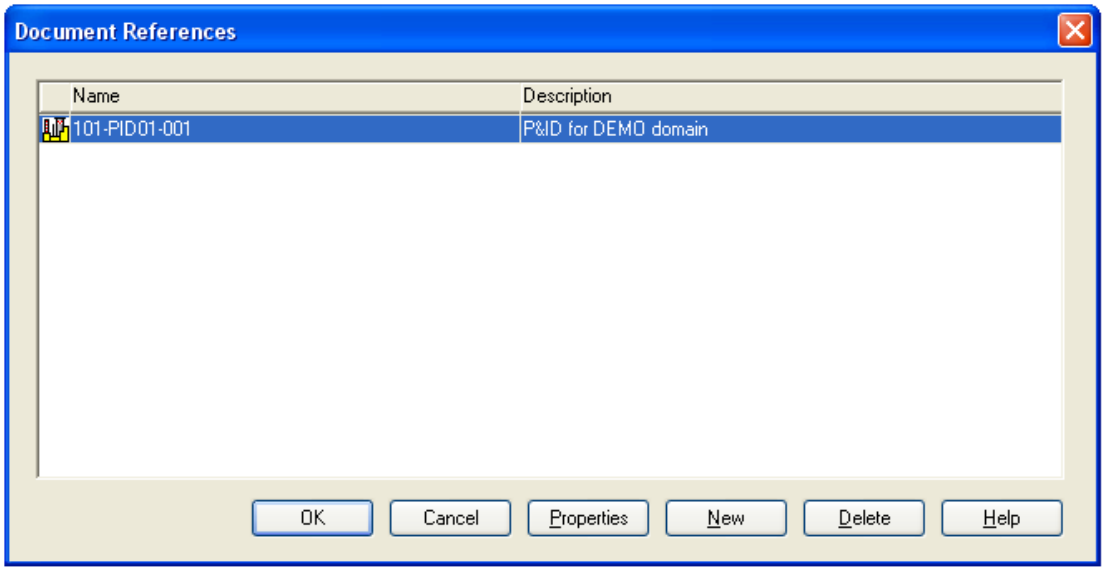
NOTE Unless you redefine the default border, the border you select in this dialog box will be used from now on for all the loop drawings you generate.

Task 5 — Editing the Title Block Data for a Specific Loop Drawing

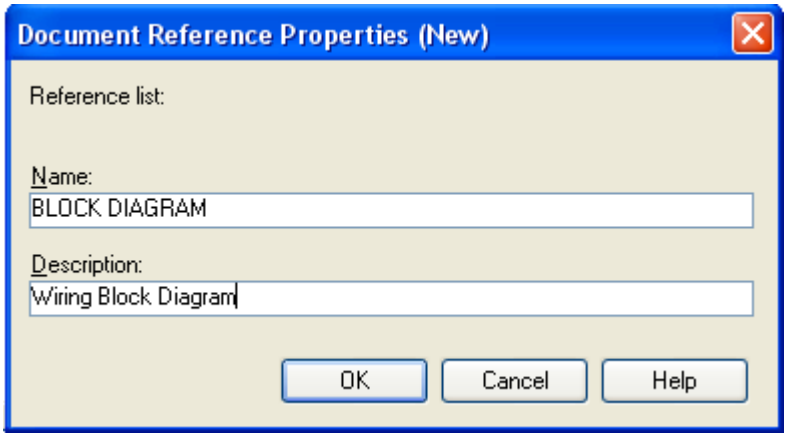
Define Drawing Title Block Information

1. In the **Domain Explorer**, select loop **101-F- 100**.
2. Right-click the loop and on the shortcut menu, click **Actions** > **Loop Drawing List**.
3. On the **Loop Drawing List** dialog box, click **References** to view the drawing reference used.

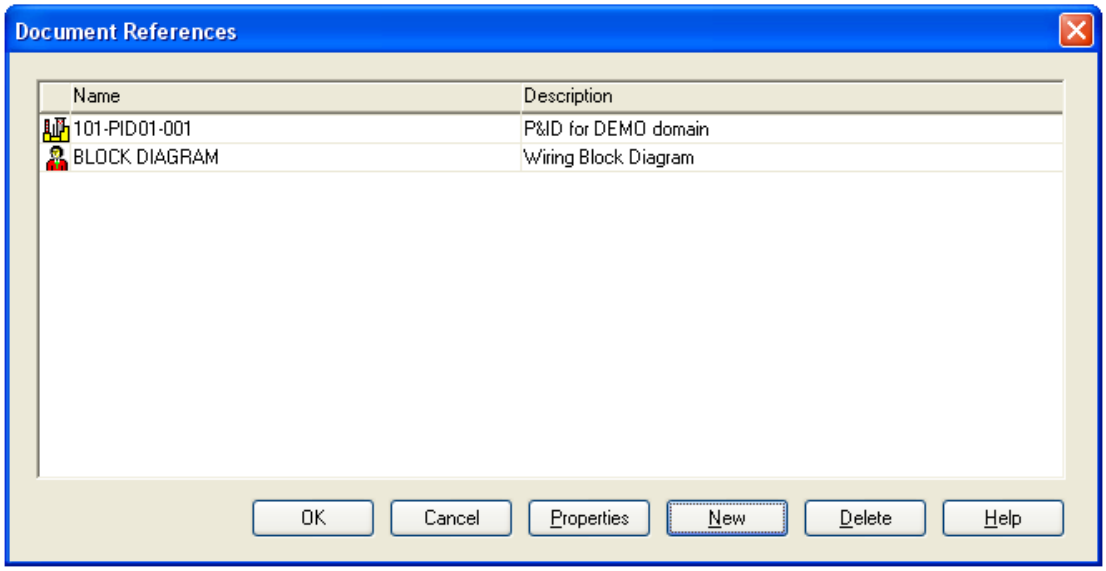
TIP On the **Document References** dialog box, the P&ID drawing reference appears automatically.



4. Click **New** to add another document reference.
5. On the **Document Reference Properties** dialog box, type data as shown.



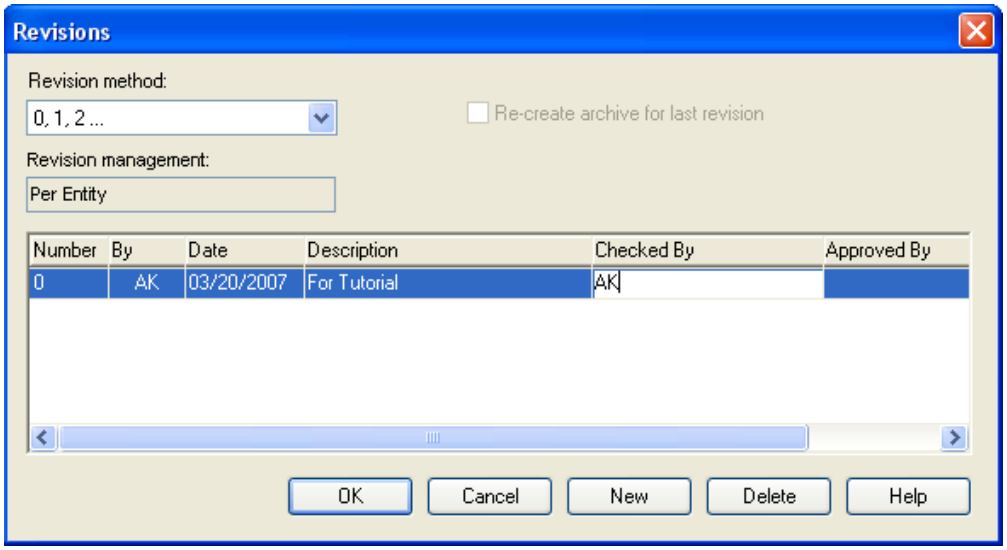
6. Click **OK** to close the dialog box and return to the **Document References** dialog box.



7. Click **OK** to close the **Document References** dialog box.

Edit Revisions

1. On the **Loop Drawing List** dialog box, click **Revisions**.
2. On the **Revisions** dialog box, add a revision as follows:
 a. In the **Revision method** list, select **0, 1, 2 …** and click **New**.
 b. Enter data similar to that shown.



3. Click **OK** to close the **Revisions** dialog box.
4. Click **Close** to close the **Loop Drawing List**.

Task 6 — Macros in CAD Loop Drawings

Macros are required to extract specific data from a given loop or instrument. You can display and modify macro definitions and define functions which you can attach to the macros and then extract the necessary string attributes.

Display Macro Definitions

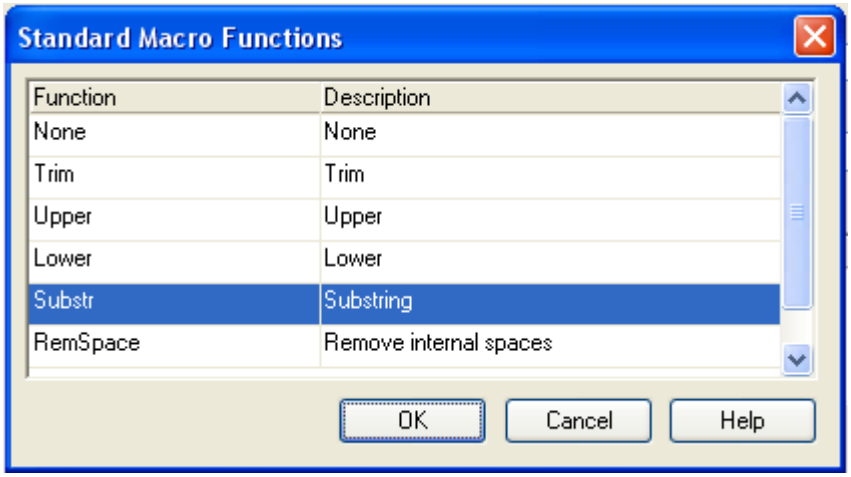
1. Do one of the following to open the **Macro Definitions** dialog box:
 - On the **Loop Drawings Module** window toolbar, click .
 - On the **Loop Drawings Module** window menu bar, click **Tables** > **Macro Definitions**.
2. Navigate to the instrument tag macros as follows:
 a. In the **Select item type** list, select **INSTRUMENT**.
 b. In the **Select item relation** list, select **INSTRUMENT - Loop**.
 c. In the **Select item property** list, select **Loop Number**.
3. In the **Macro definitions** pane, browse through the macros to see how they are organized.
4. Click **OK** to close the **Macro Definitions** dialog box.

NOTE Wiring macros include group level and wire sequence identifiers. For a fuller explanation of macros, see the *SmartPlant Instrumentation User's Guide*, *Loop Drawings Module*, *SmartPlant Instrumentation Macros*.

Define Macro Functions

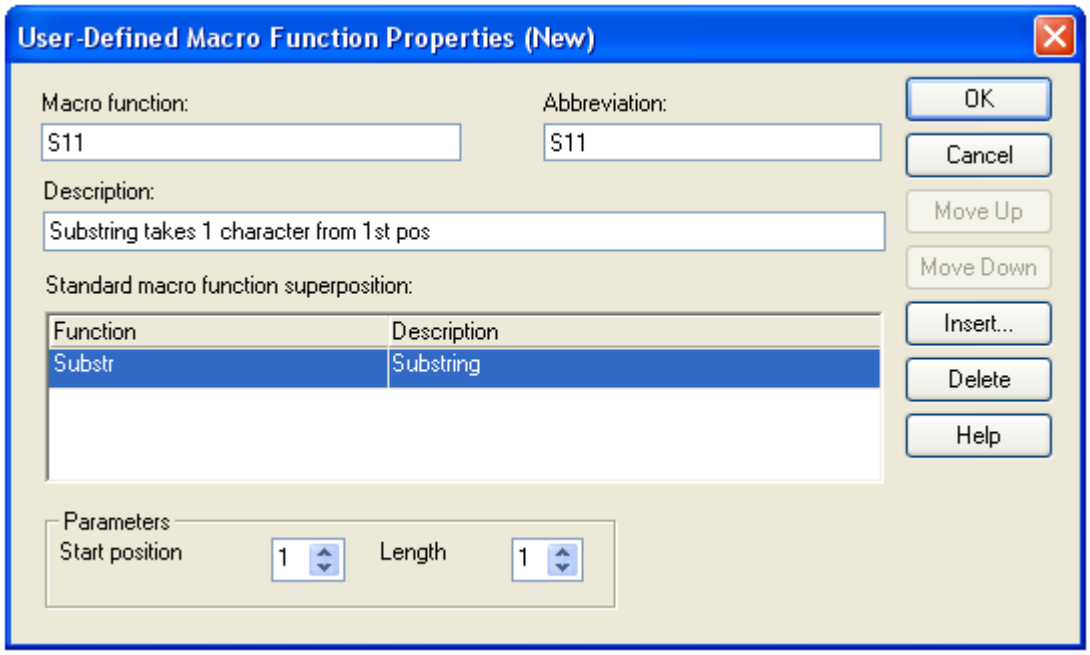
1. On the **Loop Drawings Module** window menu bar, click **Tables** > **User-Defined Macro Functions**.
2. On the **User-Defined Macro Functions** dialog box, click **New** to open the **User-Defined Macro Function Properties** dialog box.
3. Do the following:
 a. In the **Macro function** box, type **S11**.
 b. In the **Abbreviation** box, type **S11**.
 c. In the **Description** box, type **Substring takes 1 character from 1st pos**.
4. Click **Insert** to open the **Standard Macro Functions** dialog box.

5. Select the following standard function as shown.



6. Click **OK** to return to the **User-Defined Macro Function Properties** dialog box.
7. In the **Parameters** section, define the parameters as follows:
 a. In the **Start position** field, use the spin box to select the value **1**.
 b. In the **Length** field, use the spin box to select the value **1**.

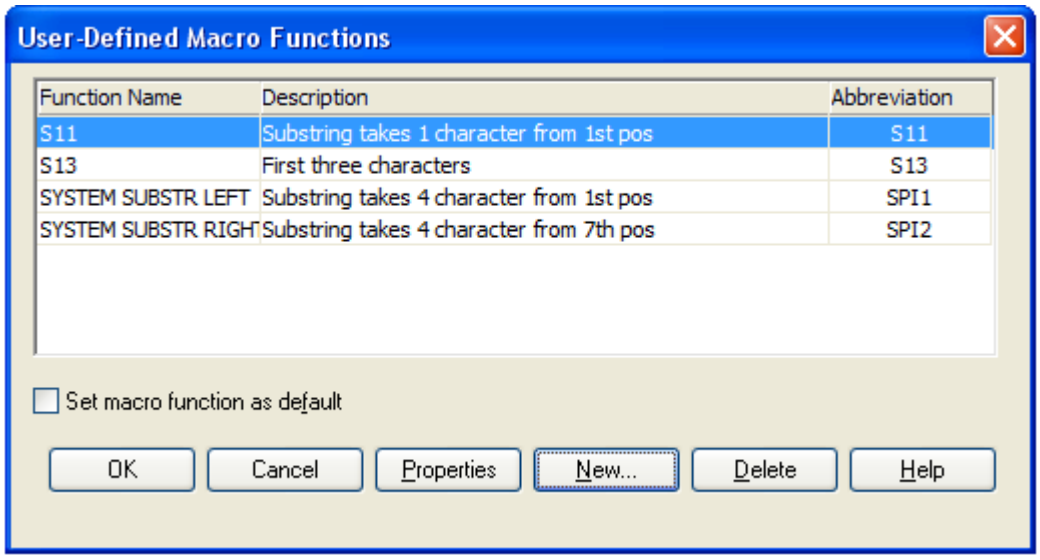
 The dialog box should now appear as shown:



8. Click **OK**.
9. Add another user-defined function **S13**, making the following definitions on the **User-Defined Macro Function Properties** dialog box:
 a. In the **Macro function** field, type **S13**.
 b. In the **Abbreviation** field, type **S13**.
 c. In the **Description** field, type **First three characters**.
 d. Click **Insert**.

e. On the **Standard Macro Functions** dialog box, select the **Substr** function.
f. Click **OK** to return to the **User-Defined Macro Function Properties** dialog box.
g. In the **Start position** field, enter **1**.
h. In the **Length** field, enter **3**.

The **User-Defined Macro Functions** dialog box should appear as shown:



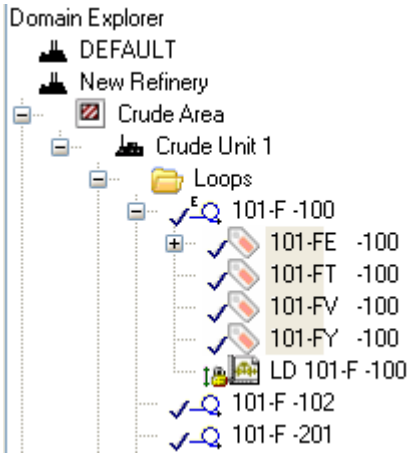
10. When done, click **OK** to close the dialog box.

For further details of how to use these functions in your drawing macros, see the *SmartPlant Instrumentation User's Guide*, *Loop Drawings Module*, *Creating User-Defined Macro Functions*.

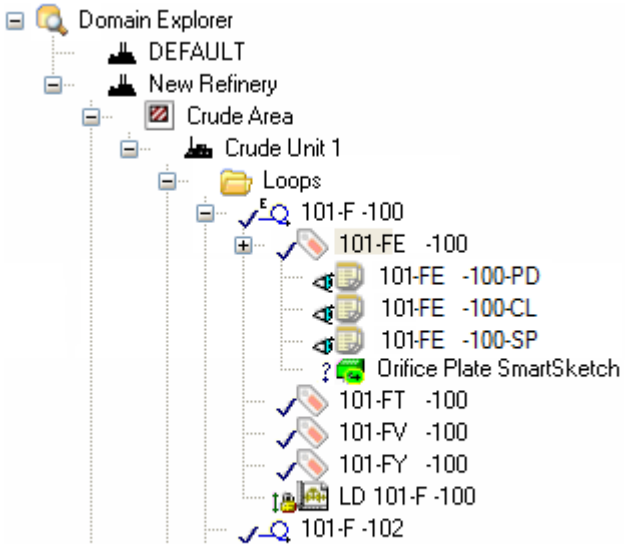
Task 7 — Viewing Assigned Blocks and Using different Block Assignment Methods

View a Drawing Block in a CAD Application

1. In the **Domain Explorer**, expand loop **101-F -100** by double-clicking its icon.
 The software displays all the tag numbers associated with this loop:



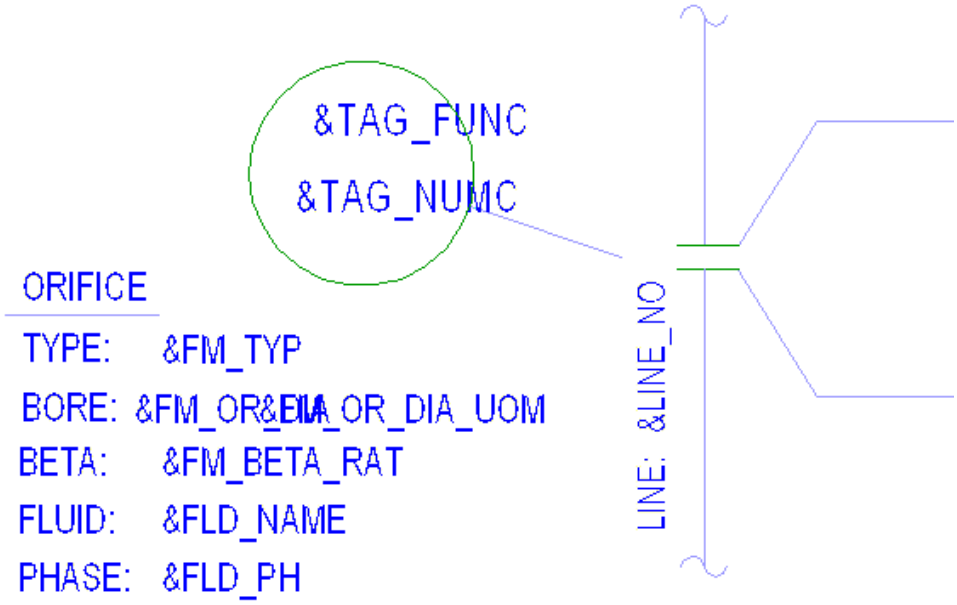
2. Double-click tag **101-FE -100** to display the drawing blocks associated with it.



TIP The **Orifice Plate SmartSketch** block is indicated with a green icon because you associated this block with the tag number using the automatic block assignment method (via instrument type).

3. Right-click the block, and on the shortcut menu, click **Reports** > **View Block in CAD Application**.

The drawing block for the selected tag is displayed in your CAD application, including the macros. (The example shows how the drawing appears in SmartSketch – what you see may be slightly different depending on the CAD application you are using).



4. Expand each of the tags in loop **101-F -100** and observe all the blocks that you associated previously.

 TIP The view obtained is in accordance with the hierarchy level selected. If you select a loop, the software displays all the drawing blocks for that loop. If you select a tag number, the software displays the blocks attached to that tag number; selection of a block displays that block only. When viewing drawing blocks before generation, the software displays the blocks with macros only.

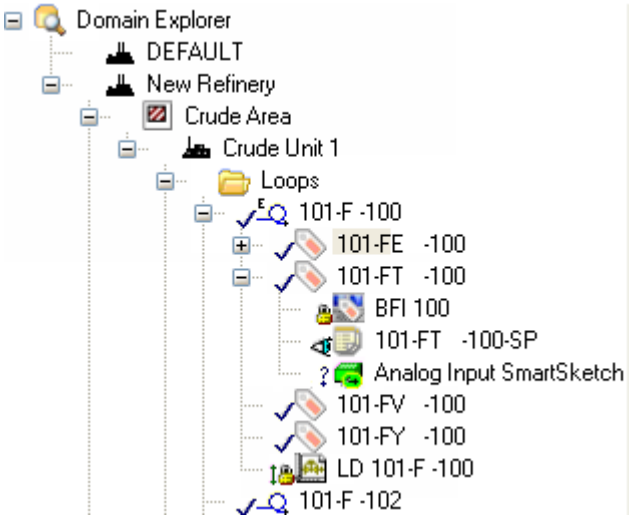
5. In the **Domain Explorer**, select loop **101-F -100**.
6. Right-click the loop, and on the shortcut menu, click **Reports** > **View in CAD Application** to view the drawing blocks attached to the loop.

Assign a Block with a Tag Using the Manual Block Assignment Method

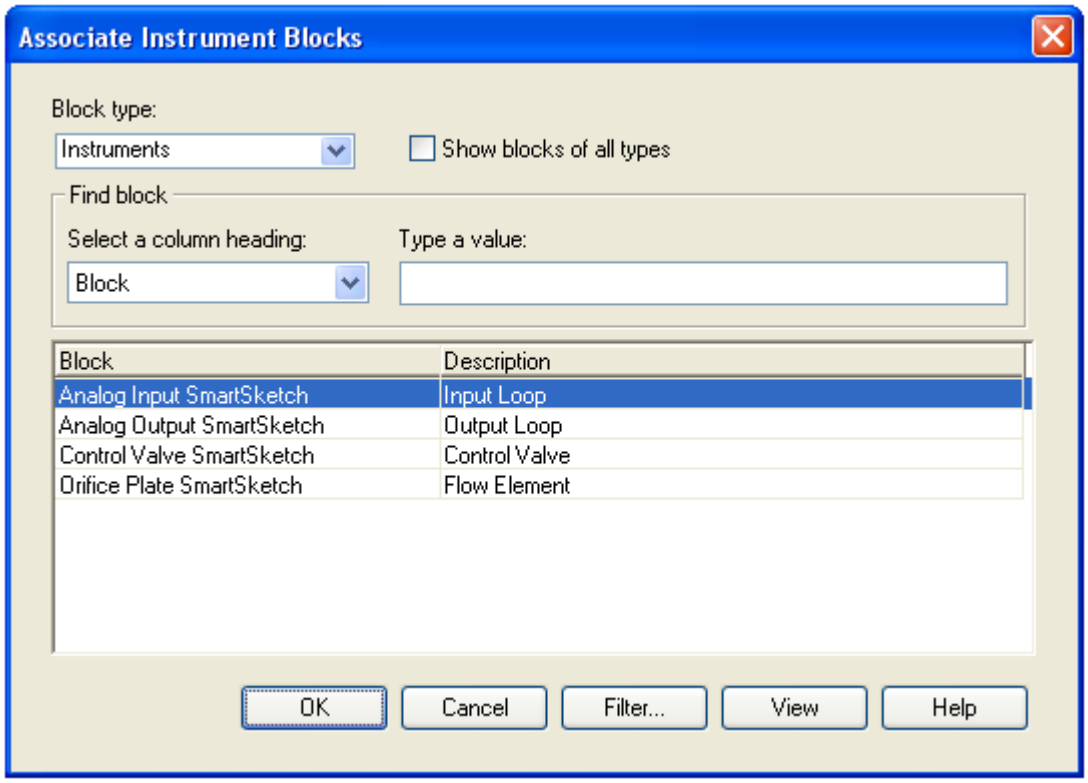
It is possible to associate instrument blocks with specific instrument tags manually. In the **Domain Explore**r, manually associated blocks are indicated with the red icon.

1. In the **Domain Explorer**, double-click loop **101-F -100** to display the tag numbers.

2. Double-click tag **101-FT -100** to display the automatically associated block.

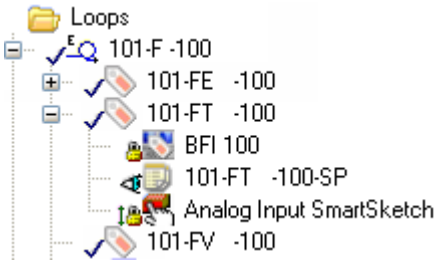


3. Right-click the **Analog Input SmartSketch** block.
4. On the shortcut menu, click **Actions** > **Dissociate Instrument Blocks** to dissociate the block.
5. When prompted, click **Yes**.
6. Right-click the **101-FT -100** tag.
7. On the shortcut menu, click **Actions** > **Associate Instrument Blocks**.
8. On the **Associate Instrument Blocks** dialog box, select block **Analog Input SmartSketch** as shown.

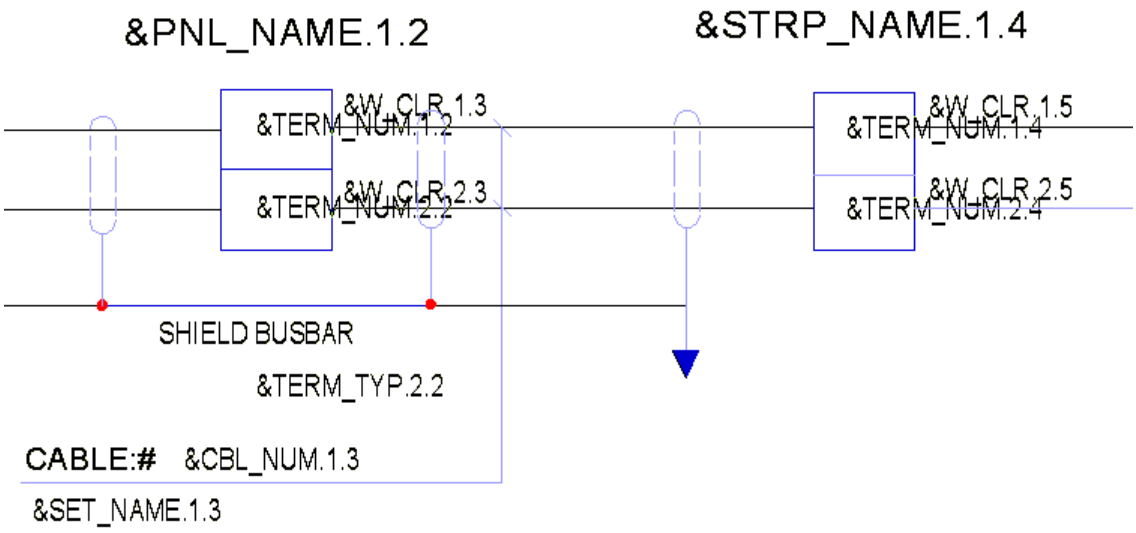


9. Click **OK**.

 In the **Domain Explorer,** you now have one red block associated with tag 101-FT -100.

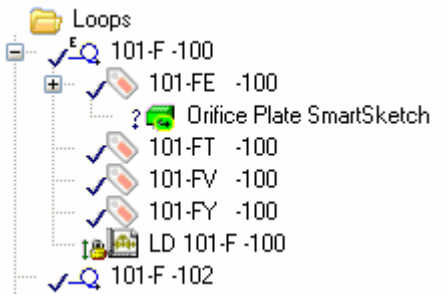


10. Right-click the **Analog Input SmartSketch** block assigned to tag **101-FT -100**.
11. On the shortcut menu, click **Reports** > **View Block in CAD Application** to display the block drawing in your CAD application.
12. Check that the wiring macros show the correct group level and sequence for the wires, as in the following example.



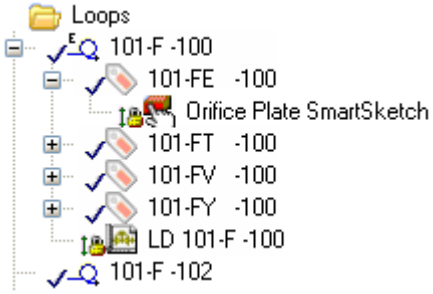
Change the Block Assignment Method from Automatic to Manual

1. In the **Domain Explorer**, double-click tag **101-FE -100** to display the automatically associated block, indicated by the icon .



2. Right-click the tag, and on the shortcut menu, click **Actions** > **Change Block Assignment Method**.
3. When prompted, click **Yes**.

 The software changes the association method to manual. In the **Domain Explorer**, the block is now indicated by the icon :



4. Repeat the same procedure for tags **101-FY -100** and **101-FV -100**.

Task 8 — Generating CAD Loop Drawings

Generate a CAD Loop Drawing

1. In the **Domain Explorer**, select and then right-click loop **101-F -100**.
2. On the shortcut menu, click **Reports** > **Generate Loop Drawings**.
3. On the **Generate Loop Drawings** dialog box, do the following:
 a. In the **Generation method** list, select **CAD**.
 b. Select the **Save drawing automatically** check box, and click **Browse** to navigate to the desired output folder.
4. Click **OK** to generate the loop drawing.
5. Inspect the results and verify that the loop drawing output reflects the wiring data as you executed in the session on the Wiring module.

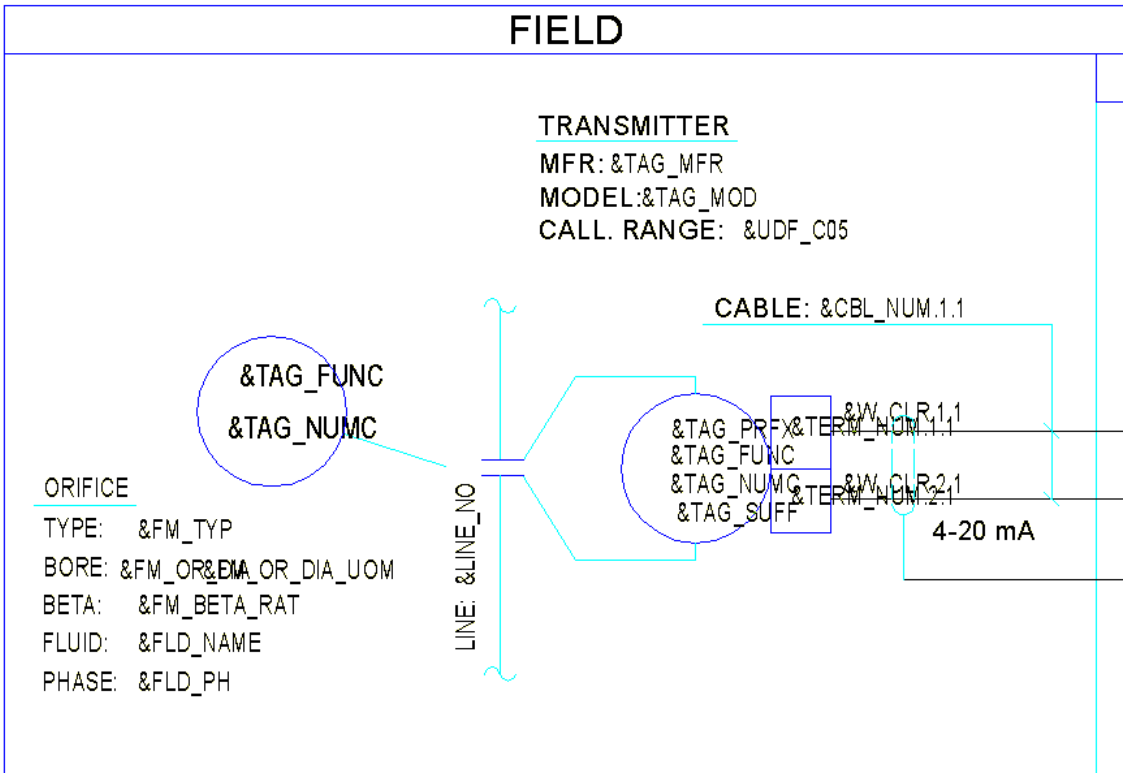
Add a Revision to a CAD Loop Drawing

1. In the **Domain Explorer**, select and then right-click loop **101-F -100**.
2. On the shortcut menu, click **Actions** > **Loop Drawing List**.
3. Click **Revisions**.
4. On the **Revisions** dialog box, click **New** to add a new a revision for the loop.
5. Regenerate the loop and view the new revision data in the loop drawing.

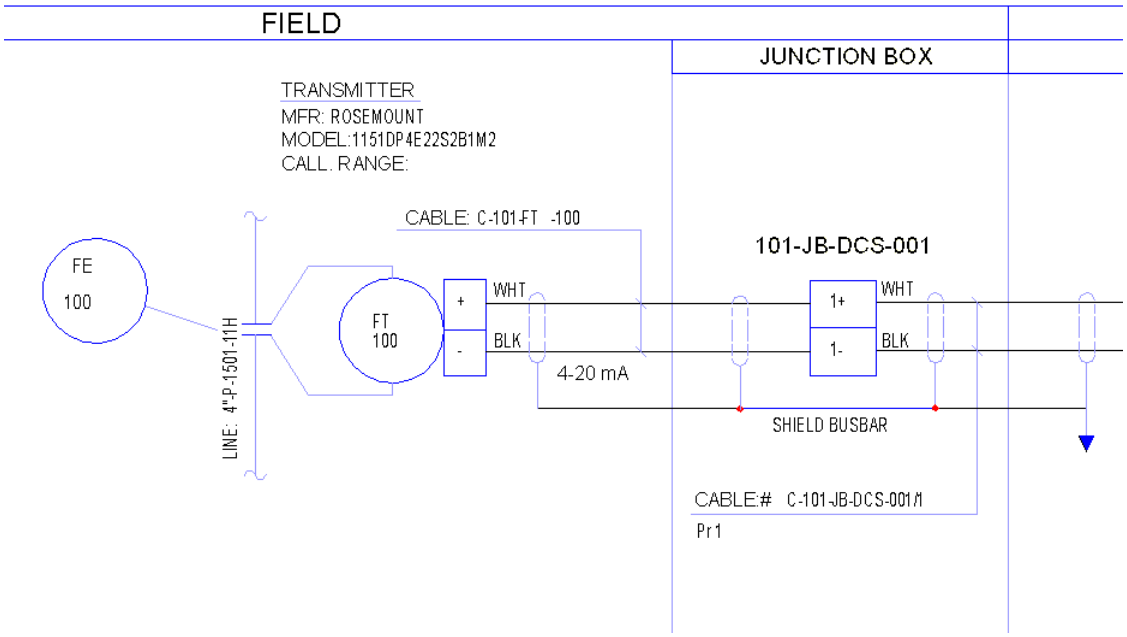
NOTES

- The following screen captures show how the loop drawing for loop **101-F -100** appears before and after generation. Note how the macros are replaced by SmartPlant Instrumentation data after the generation.

CAD Loop Drawing with Macros Before Generation



CAD Loop Drawing After Generation



SECTION 12

Working with Hook-Ups

In this section, you are going to use the **Reference Explorer** and **Domain Explorer** options to create, edit, and view the physical installation details (hook-ups) of instruments. After making all the necessary associations, you will also generate hook-up drawings using the Enhanced Report Utility, and also generate a Bill of Material.

When generating an enhanced hook-up drawing, the software uses the following elements:

- Layout file (you define the default layout on the **Preferences** dialog box).
- Title block file (you define the default layout on the **Preferences** dialog box).
- A symbol with macros (you associate the appropriate symbol file with each hook-up when defining hook-up properties).

Task 1 — Setting Preferences for Enhanced Hook-Up Drawings

Enhanced hook-up drawings are drawings you generate using the Enhanced Report Utility, which is part of SmartPlant Instrumentation software package.

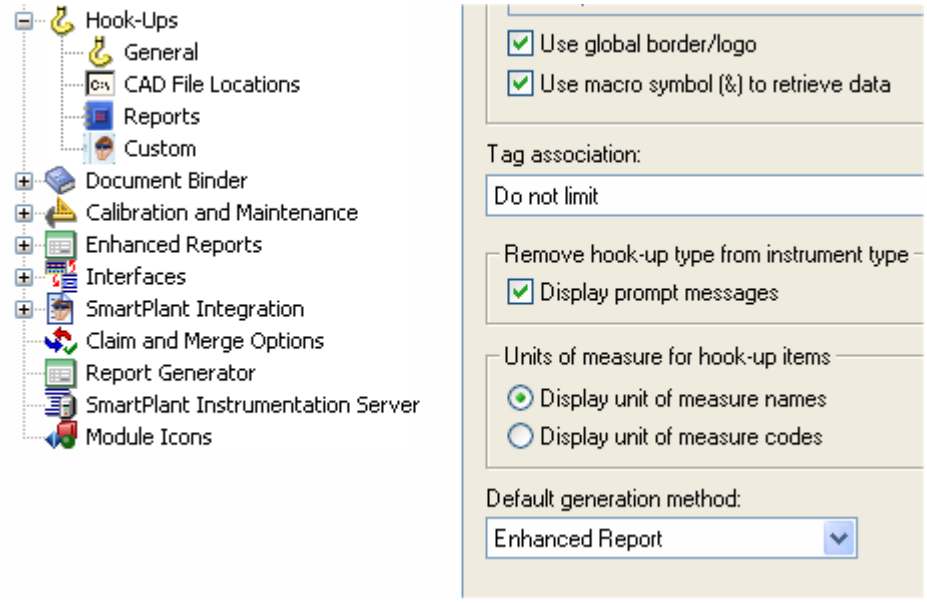
In this task, you will set the Enhanced Report Utility as your default hook-up drawing generator. This preference enables you to open the shipped .sym files from SmartPlant Instrumentation and view the hook-up item and instrument tag macros that these .sym files contain. After assigning a .sym file to a hook-up, you can generate an enhanced hook-up drawing.

Also, you define the default layout and title block files for enhanced hook-up drawings.

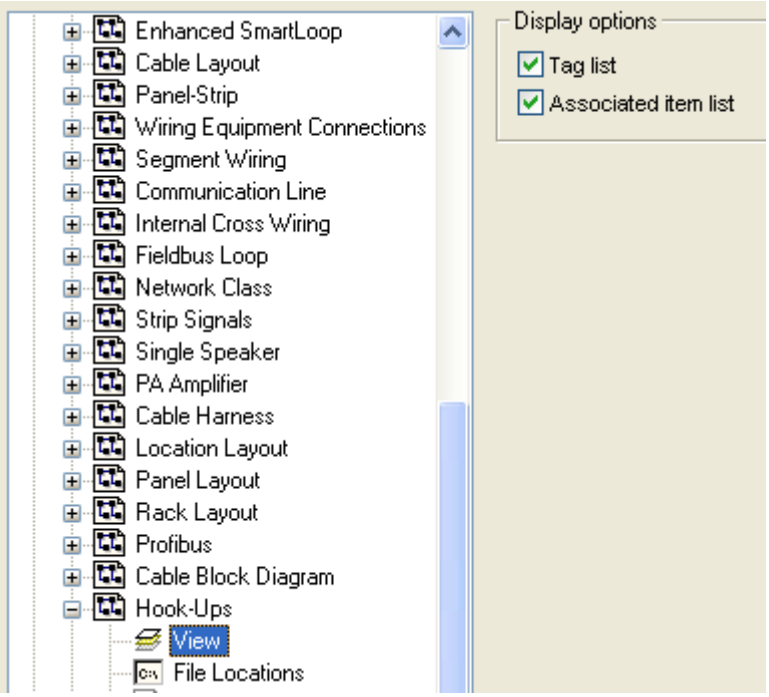
Set Preferences for Enhanced Hook-Up Drawings

1. Click **File** > **Preferences**.
2. On the **Preferences** > **General** page ensure that you have defined the **SmartPlant Instrumentation temporary folder path**.

3. On the **Hook-Ups** > **General** page, under **Default generation method**, select **Enhanced Report**.



4. On the **Enhanced Reports** > **Hook-Ups** > **View** page, from the **Display Options** select the **Tag list** and **Associated item list** check boxes.



When generating a Hook-Up drawing in the Enhanced Report Utility, selecting these options generates a tag list and an associated item list on the drawing.

5. On the **Enhanced Reports** > **Hook-Ups** > **File Locations** page, do the following:
 a. Beside **Template for default layout**, click **Browse** and specify the **A4tall.sma** file as the default layout file to be associated with your enhanced hook-up drawings at the time of generation.

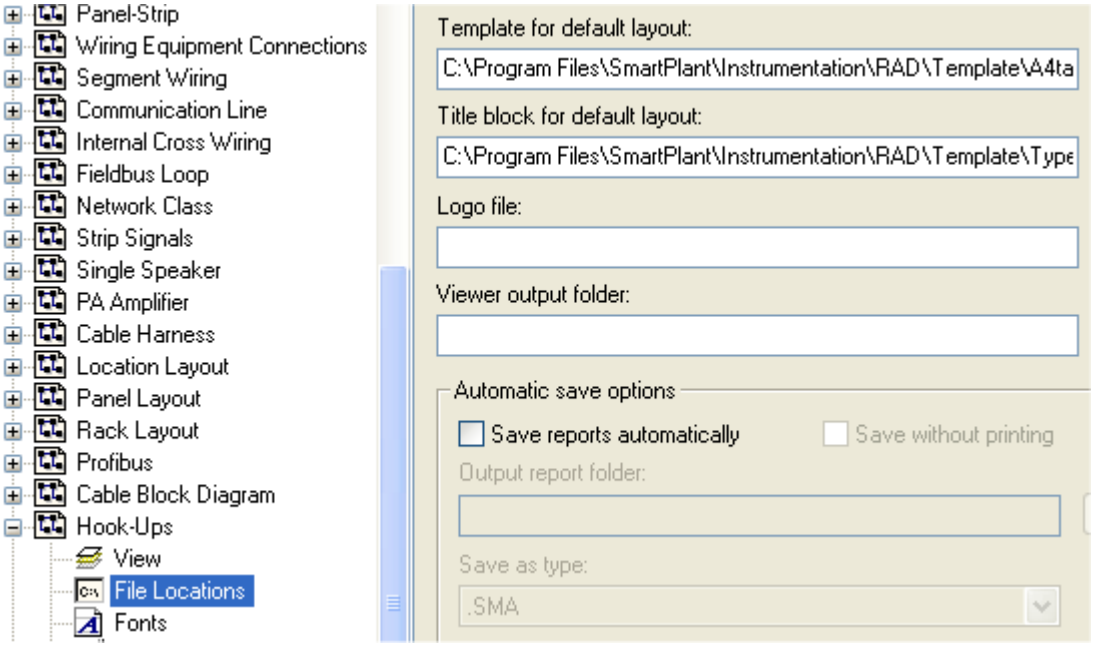
This is a shipped file, which resides in the following path:

<SmartPlant Instrumentation home folder>\RAD\Template\

b. Beside **Title block for default layout**, click **Browse** and specify the **HookUp_Tall.sym** file as the default template file to be associated with your enhanced hook-up drawings at the time of generation.

This is a shipped file, which resides in the following path:

<SmartPlant Instrumentation home folder>\RAD\Template\Types\HookUp\



6. Click **OK** to save the changes and close the **Preferences** dialog box.

Task 2 — Creating Hook-Up Libraries and Items

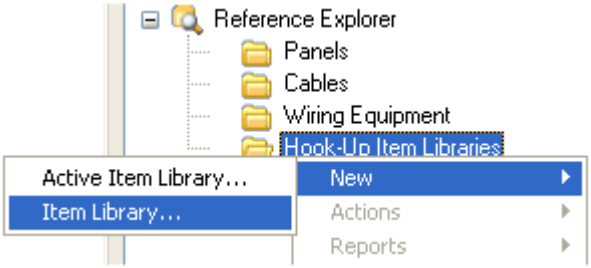
In SmartPlant Instrumentation, you can define a large number of hook-up items and group them in item libraries and sub-libraries. You create and manage item libraries and items in the **Reference Explorer**. When you create an item library, the software automatically creates a default sub-library that is contained in that item library. You can use this default sub-library for assigning hook-up items if you do not work with pipe specs. Since the Tutorial tasks do not cover pipe specs, you will be defining hook-up items in the default sub-library.

In this task, you first create hook-up libraries, set one of the libraries as active, and then define items in the active library. The active item library is your default library for the current plant. After you add hook-up items in this library, you can associate specific items with the hook-ups that you created in Task 2. The associated items can then appear in a Bill of Material and enhanced hook-up drawings that you generate.

When defining a hook-up item, you can enter a number of hook-up item properties, such as item size, manufacturer, material, storage number, unit of measure, and so forth.

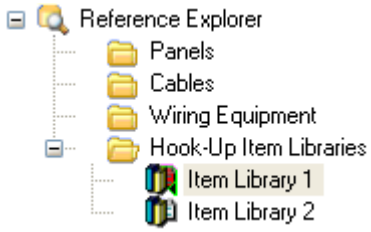
Create Hook-Up Item Libraries and Set One Library as Active

1. Press **F8** to open the **Reference Explorer**.
2. Right-click **Hook-Up Item Libraries** and then, on the shortcut menu, click **New** > **Item Library**.



3. On the **Item library Properties** dialog box, under **Item library**, enter **Item Library 1**, and then, click **OK**.
4. Create another library **Item Library 2**.
5. In the **Reference Explorer**, right-click **Item Library 1** and then, on the shortcut menu, click **Actions** > **Set as Active Item Library**.

 In the **Reference Explorer**, the libraries should now appear as shown (note the differences between the library icons):



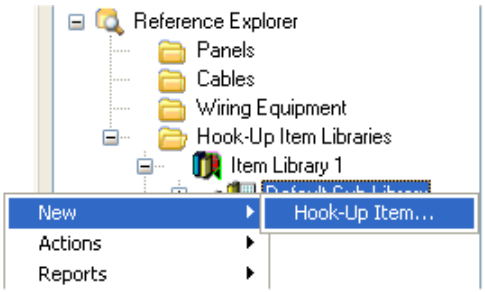
 TIP It is also possible to define a library as the active item library on the fly, by selecting the **Hook-Up Item Libraries** folder and using the **New** > **Active Item Library** command on the shortcut menu. You can only set one specific item library as the active item library for your plant.

Create Hook-Up Items for Associating with the Control Valve Hook-Up

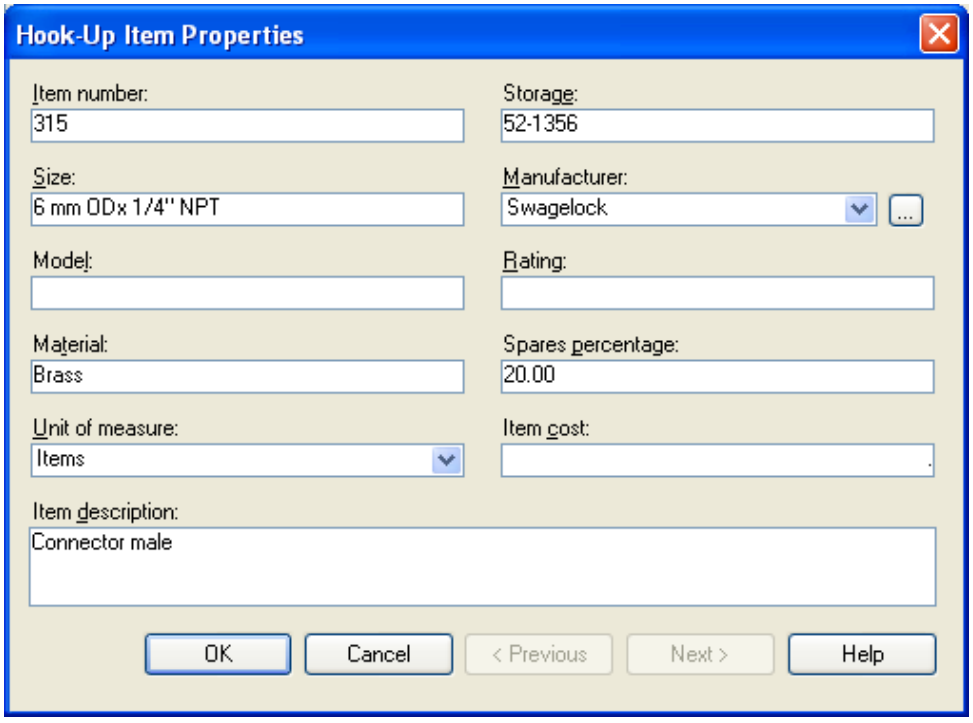
In this procedure, you create hook-up items that appear on the drawing symbol associated with the **Control Valve** hook-up.

1. In the **Reference Explorer**, double-click **Item Library 1** to display its default sub-library.

2. Right-click **Default Sub-Library** and then, on the shortcut menu, click **New** > **Hook-Up Item**.



3. On the **Hook-Up Item Properties** dialog box, enter data as shown.



TIPS

- To add manufacturers, click the ellipsis button  beside the **Manufacturer** property, and then, on the **Item Manufacturers** dialog box, add new manufacturers **Swagelock** and **Worcester**.
- The spares percentage value is taken into account when SmartPlant Instrumentation calculates the item total in the Bill of Material.

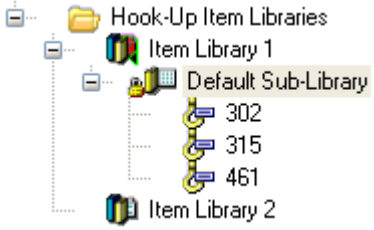
4. Click **OK**.
5. Create two more hook-up items with the following properties.

Number	Size	Material	Spares %	Unit	Description
302	6 mm OD	CU	10.00	Meter	Tubing
461	1/4" NPT	Brass	20.00	Items	Ball valve

TIPS

- Use Steps 2-5 to create a new hook-up item. Alternatively, right-click item **315** and then, on the shortcut menu, click **Duplicate** and update the new item properties for the new item.
- It is possible to create items in batch mode (right-click **Default Sub-Library**, and then, on the shortcut menu, click **Actions** > **Batch Item Creation**). However, you still need to define properties individually for each item that you create.

6. In the **Reference Explorer**, double-click **Default Sub-Library** to display the created items.



Create Hook-Up Items for Associating with the Flow Instrument Hook-Up

In this procedure, you create hook-up items that appear on the drawing symbol associated with the **Flow Instrument** hook-up.

1. In the **Reference Explorer**, double-click **Item Library 1** to display its default sub-library.
2. Using the hook-up item creation options, create hook-up items that have the following properties.

Number	Size	Manufacturer	Material	Unit	Description
25	1/2" x 2"	Swagelock	CS	Items	Nipple
32	1/2"	Swagelock	CS	Items	Elbow
45	1/2"	Swagelock	GCS	Items	Plug
149	1/2"	Worcester	SS	Items	Ball valve
172	1/2" OD	Swagelock	SS	Meter	Tube
303	1/2" OD x 1/2" NPT	Swagelock	SS	Items	Connector male
305	1/2"	Swagelock	CS	Items	Tee equal seamless
841	1/2" OD x 1/2"	Swagelock	SS	Items	Tee run male

Generate a Report of Hook-Up Items

In this procedure, you generate a report containing all the created items and the item properties.

1. In the **Reference Explorer**, in the **Hook-Up Item Libraries** folder, right-click **Item Library 1**.
2. On the shortcut menu, click **Reports** > **Library Items**.

3. When prompted to open the report print preview, click **Yes**.

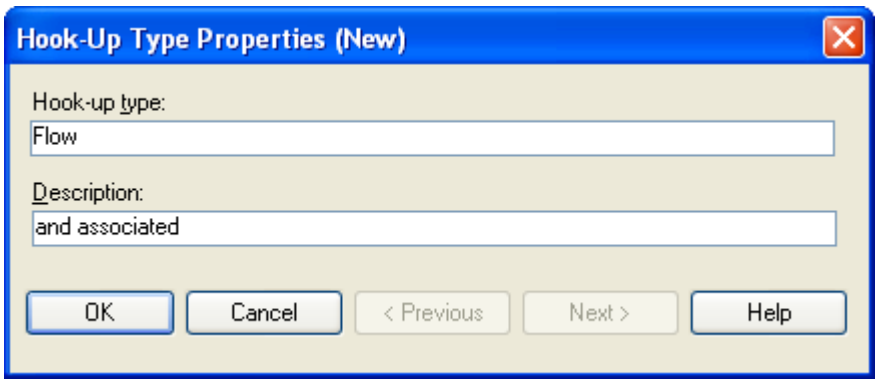
Task 3 — Creating Hook-Ups and Assigning them to Instruments

In this task, you are going to use the **Domain Explorer** to create hook-up types and hook-ups and assign these hook-ups to instruments. In the **Domain Explorer**, hook-up types and hook-ups are displayed at the plant level.

Also, you will associate a .sym (symbol) file with each hook-up. A symbol file contains an instrument installation sketch with hook-up item macros and instrument tag macros. After associating a .sym file with a hook-up, you can open this file in the Enhanced Report Utility to view the macros before generating the enhanced hook-up drawing.

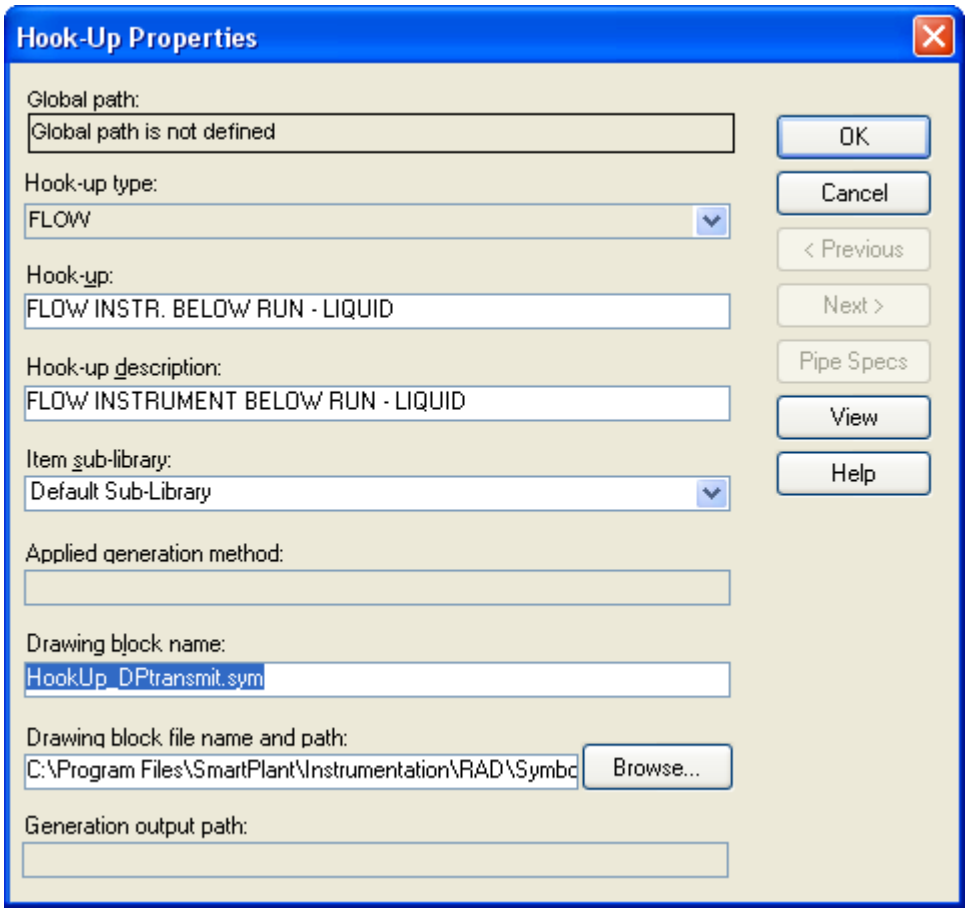
Create Hook-Up Types and Hook-Ups

1. Press **F7** to open the **Domain Explorer** and double-click **New Refinery** to display folders that appear at the plant level.
2. Right-click the **Hook-Ups** folder and then, on the shortcut menu, click **New** > **Hook-Up Type**.
3. On the **Hook-Up Type Properties** dialog box, type data as shown.



4. Click **OK** to display the **Flow** hook-up type in the **Domain Explorer**.
5. Right-click the hook-up type you just created and then, on the shortcut menu, click **New** > **Hook-Up**.

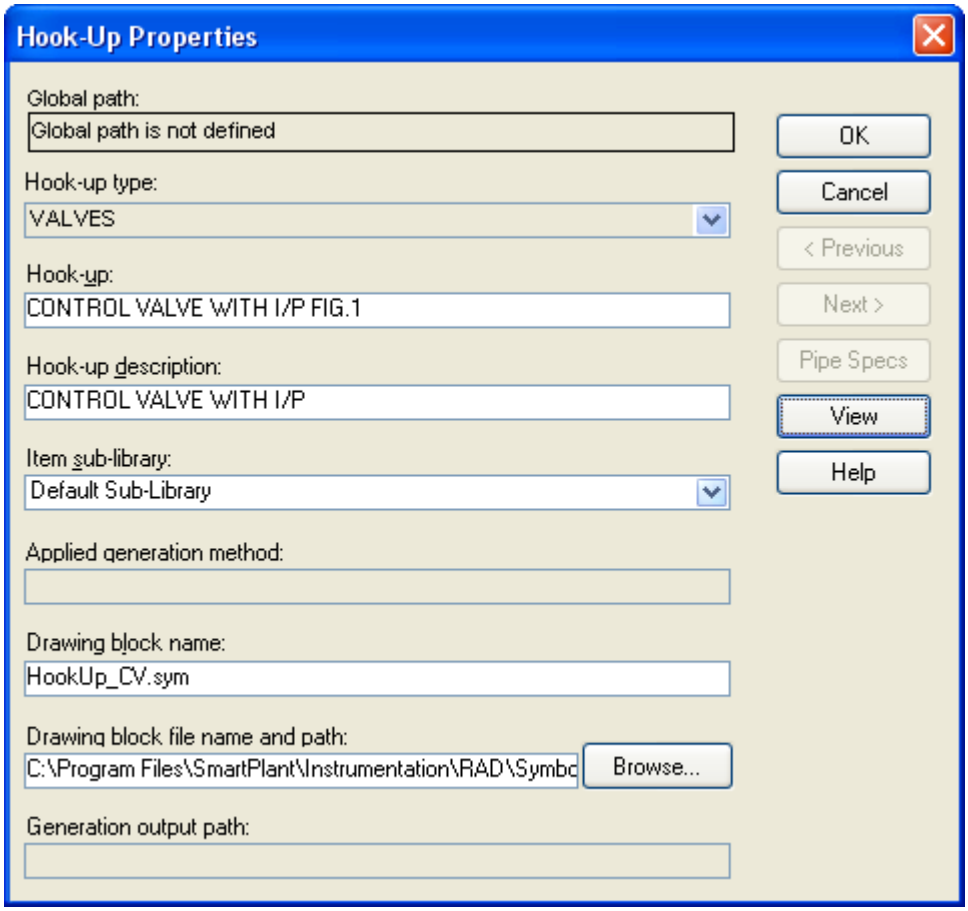
6. On the **Hook-Up Properties** dialog box, type the hook-up name and description as shown, and then, click **Browse** to specify the symbol file **HookUp_DPtransmit.sym** in the path <SmartPlant Instrumentation home folder>\RAD\Symbols\Custom.



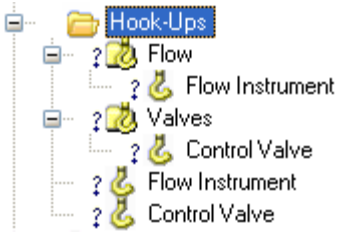
IMPORTANT Only symbols of file type .sym can be used when generating an enhanced hook-up drawing.

7. Create another hook-up type called **Valves** with the **All types** description.

8. Under the **Valves** hook-up type, create a hook-up called **Control Valve** with the **Control valve with I/P** description and assign the **HookUp_CV.sym** file to it.



On completion, the hook-up types and hook-ups should appear in the **Domain Explorer** as shown:

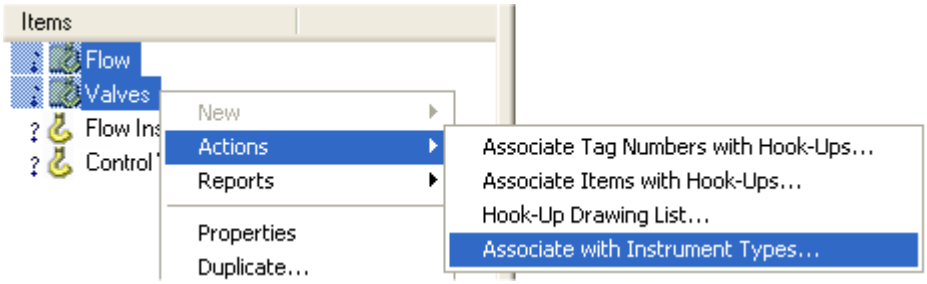


Associate Hook-Up Types with Instrument Types

1. In the **Domain Explorer**, click the **Hook-Ups** folder to displays all hook-up types and hook-ups in the **Items** pane.

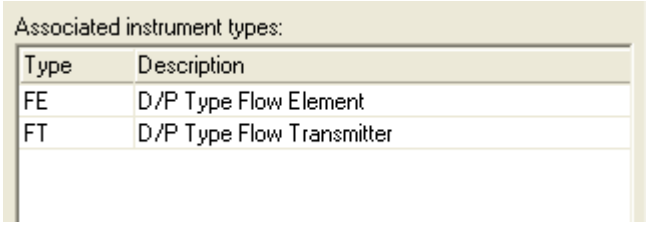


2. Select the **Flow** and **Valves** hook-up types and then, on the shortcut menu, click **Actions** > **Associate with Instrument Types**.



3. On the **Associate Hook-Up Types with Instrument Types** dialog box, select instrument type **FE – D/P Type Flow Element**.
4. Do one of the following to associate the instrument type with the **Flow** hook-up type:
 - Click **Associate**.
 - Drag the **FE – D/P Type Flow Element** instrument type to the **Associated instrument types** pane.
5. In the same way, associate instrument type **FT – D/P Type Flow Transmitter** with the **Flow** hook-up type.

 The **Associated instrument types** pane should appear as shown:

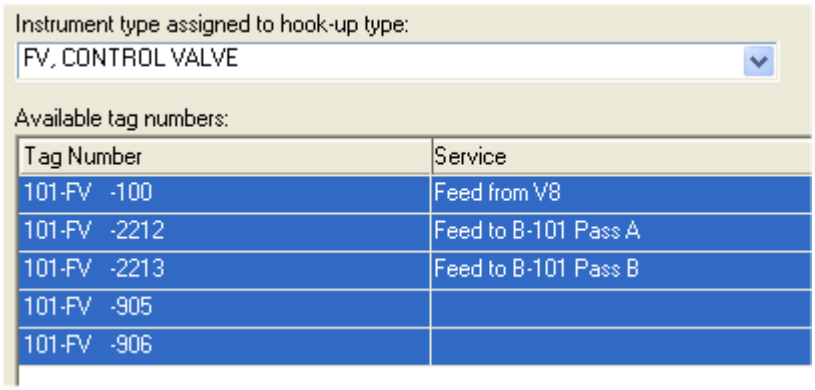


6. Click **Next**.
7. Associate instrument type **FV – CONTROL VALVE** with the **Valves** hook-up type.
8. Click **OK**.

Associate Instrument Tags with Hook-Ups

1. In the **Domain Explorer**, double-click the **Hook-Ups** folder and right-click the **Control Valve** hook-up.
2. On the shortcut menu, click **Actions** > **Associate Tag Numbers with Hook-Ups**.

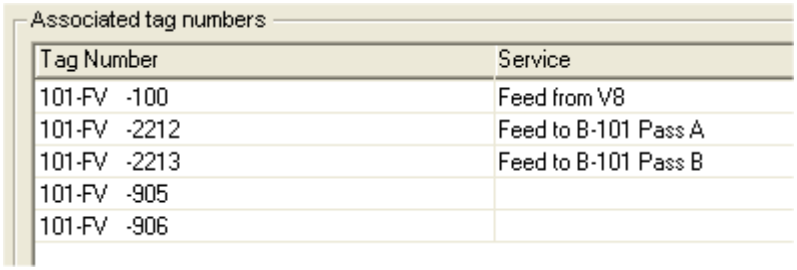
 TIP The dialog box displays all instruments whose instrument type is you assigned to the **Valves** hook-up type (that is, **FV – CONTROL VALVE**).
3. From the **Instrument type assigned to hook-up type** list, select **FV, CONTROL VALVE**.
4. Under available tag number, select the following control valve tags.



5. Click **Associate**.

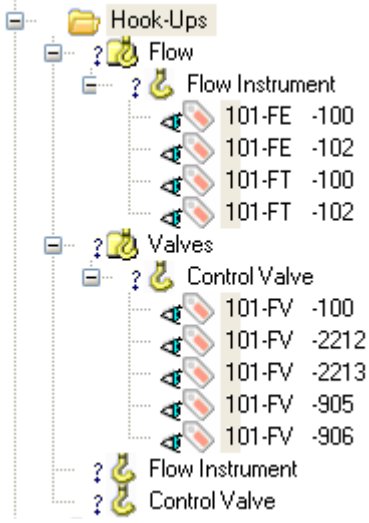
 TIP You can also associate tags by dragging them one by one to the **Associated tag numbers** pane.

 The **Associated tag numbers** pane should appear as shown:



6. Click **OK**.
7. Now associate flow instrument tags **101-FE -100**, **101-FE -102**, **101-FT -100**, and **101-FT -102**, with the **Flow Instrument** hook-up.

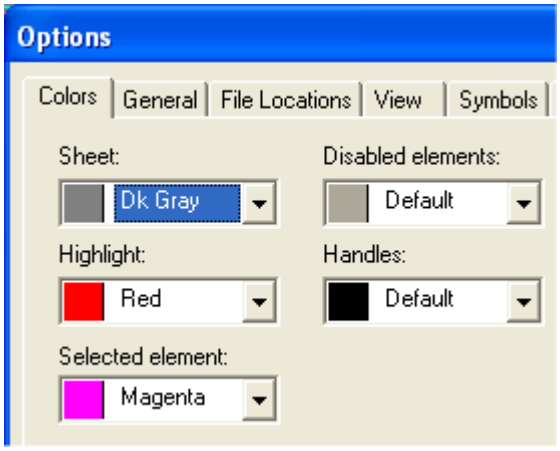
In the in the **Hook-Ups** folder of the **Domain Explorer**, the associated instruments should appear as shown:



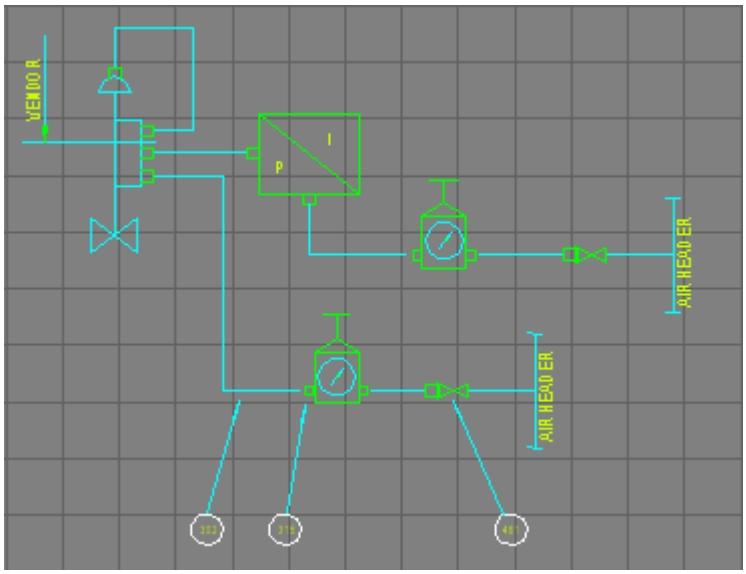
NOTE In the **Hook-Ups** folder, instruments assigned to hook-ups appears as view-only, which is indicated by the icon.

View Drawing Symbol Content

1. In the **Domain Explorer**, click the **Hook-Ups** folder to display its content in the **Items** pane.
2. In the **Items** pane, select and right-click the **Flow Instrument** and **Control Valve** hook-ups.
3. On the shortcut menu, click **Properties**.
4. On the **Hook-Up Properties** dialog box, click **View** to display the **Control Valve** hook-up associated symbol file, **HookUp_CV.sym** in the symbol editor.
5. Click **Tools** > **Options**, and then, under **Sheet**, select **Dk Gray** (dark gray).



The symbol should appear as shown:



TIP The symbol displays the following items: **302** (tubing), **315** (two connector males), and **461** (one ball valve). You are going to create these items in the next task (Task 3).

6. On the **Hook-Up Properties** dialog box, click **Next**.
7. Click **View** to display the symbols for the **Flow Instrument** hook-up.

Task 4 — Associating Items with the Hook-Ups

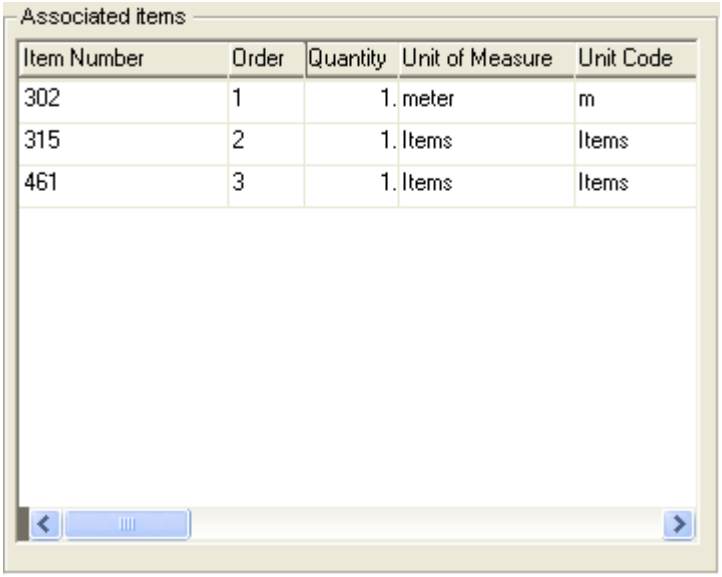
So far, you have created hook-up items in the **Reference Explorer** and hook-ups in the **Domain Explorer**. Now you need to associate your items with the appropriate hook-ups so that you can then display these items in a Bill of Material and enhanced hook-up drawings.

When associating items with hook-ups, you define the item quantity according to the number of instruments included in the hook-up.

Assign Items from the Active Item Library to the Control Valve Hook-Up

1. Press **F7** to open the **Domain Explorer**.
2. In the **Hook-Ups** folder, right-click the **Control Valve** hook-up.
3. On the shortcut menu, click **Actions** > **Associate Items with Hook-Ups.**
4. Under **Items available for association**, select items **302**, **315**, and **416** and do one of the following:
 - Click **Associate**.
 - Drag the selected items one by one to the **Associated items** pane.

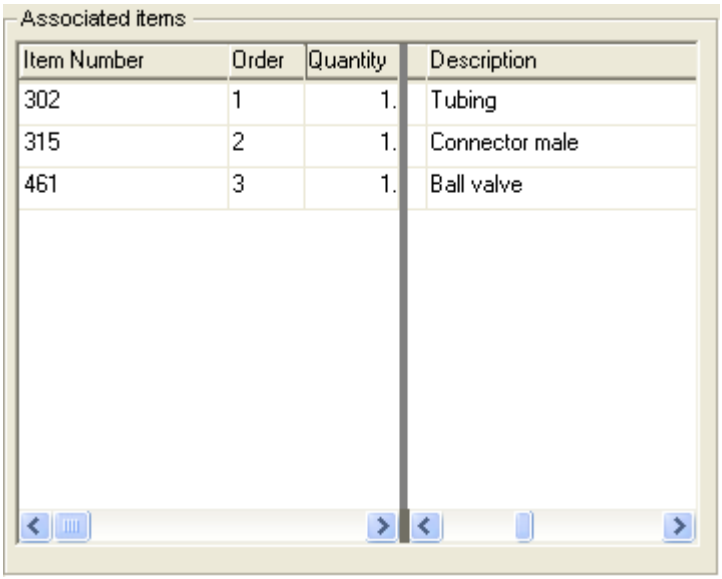
The **Associated items** pane should appear as shown:



5. View the associated items in a split display by dragging the vertical bar to the right. To find the vertical bar, place the cursor to the left of the left scroll box arrow so that the cursor changes its shape as shown.



6. Using the scroll boxes, customize the column display as shown.



7. In the **Quantity** box, enter the values for the item quantity you want to display when you generate the Bill of Material for the instruments associated with the **Control Valve** hook-up.
 a. For the **302 (Tubing)** item, enter **3**. The Bill of Material will show the total tubing length as **16.5 meters** (three meters for each control valve + 10% spares).

b. For the **315 (Connector male)** item, enter **2**. The Bill of Material will show the item total as **12** male connectors (two male connectors for each control valve + 20% spares).

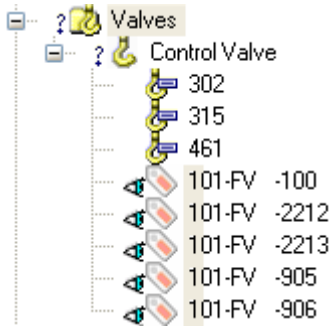
c. For the **461 (Ball valve)** item, enter **1**. The Bill of Material will show the item total as **6** ball valves (one ball valve for each control valve + 20% spares).

NOTE The values that you enter in the **Quantity** column depend on the unit of measure you defined for the item.

Item Number	Order	Quantity		Description
302	1	3.		Tubing
315	2	2.		Connector male
461	3	1.		Ball valve

8. When done, click **OK** to close the dialog box.
9. In the **Hook-Ups** folder of the **Domain Explorer**, expand the hierarchy of the **Valves** hook-up type.

 The items should appear as shown:



Assign Items from the Active Item Library to the Flow Instrument Hook-Up

1. In the **Domain Explorer**, in the **Hook-Ups** folder, right-click the **Flow Instrument** hook-up.
2. On the shortcut menu, click **Actions** > **Associate Items with Hook-Ups**.
3. Associate the hook-up numbers and define the item quantities as shown.

Number	Description	Quantity	Unit
25	Nipple	4	Items
32	Elbow	2	Items
45	Plug	4	Items
149	Ball valve	2	Items
172	Tube	4	Meter
303	Connector male	4	Items
305	Tee equal seamless	2	Items
841	Tee run male	2	Items

Task 5 — Generating a Bill of Material (BOM) and a Hook-Up Drawing

Now you are ready to generate a Bill of Material for each hook-up that you created and also generate enhanced hook-up drawings using the Enhanced Report Utility.

Generate a Bill of Material

1. In the **Domain Explorer**, in the **Hook-Ups** folder, right-click the **Control Valve** hook-up, and then, on the shortcut menu, click **Reports** > **Bill of Material**.
2. When prompted to open the report print preview, click **Yes**.
3. Close the **Domain Explorer** and **Reference Explorer** to display the Bill of Material **Print Preview** window in full screen view.
4. Under the **Total Qty** (quantity) / **For Order** column, check how SmartPlant Instrumentation calculated the total quantity for each hook-up item.

Unit of Measure Rating Cost	Total Qty For Order
meter 0	16.5
Items 0	12
Items	6

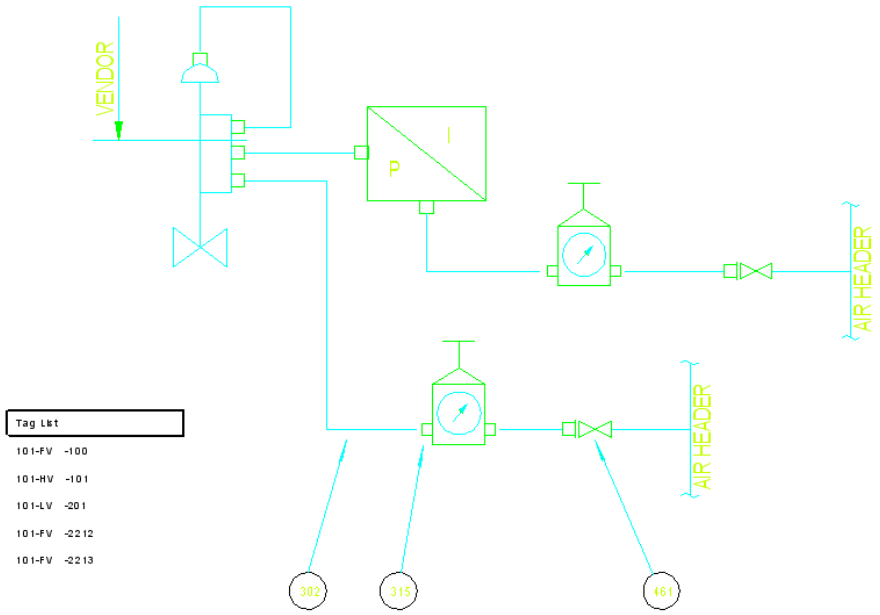
5. In the **Total Qty** / **For Order** column, if the total quantity value is greater or smaller than you require, you can enter manually the actual number of items that want to order in the **For Order** field. To do so, do the following:
 a. Select the **Edit mode** check box
 b. Under the total quantity value, type the appropriate value for order.

Total Qty For Order
16.5
17

 TIP The value that you enter in the **For Order** field only appears in the printout.
6. Now generate a Bill of Material for the **Flow Instrument** hook-up.

Generate an Enhanced Hook-Up Drawing

1. In the **Domain Explorer**, in the **Hook-Ups** folder, right-click the **Control Valve** hook-up, and then, on the shortcut menu, click **Reports** > **Generate Hook-Up Drawings**.
2. On the **Generate Hook-Up Drawings** dialog box, click **OK**.
3. When prompted to open the report print preview, click **Yes** to open the drawing in the Enhanced Report Utility.
4. Check whether the macros have retrieved the instrument tag numbers and hook-up item properties.



Number	Description	Size	Material	Rating	Type
461	BALL VALVE-CHROME	1/4" NPT	BRASS	-	VALVES
302	TUBING	6 mm OD	CU	-	VALVES
315	CONNECTOR MALE	6 mm OD x 1/4" NPT	BRASS	-	VALVES

5. Close the Enhanced Report Utility.
6. Generate an enhanced report hook-up drawing for the **Flow Instrument** hook-up.

NOTES

- Only symbols of file type .sym can be used when generating an enhanced hook-up drawing.
- The **Tag list** and **Associated items list** can be dragged to any position on the Enhanced Hook-Up Drawing, but their positions cannot be saved within the drawing, unless the drawing is saved as an external file. Printing the drawing creates the drawing with the lists as displayed in the Enhanced Report Utility.
- Unlike other drawing types generated in the **Enhanced Report Utility**, you cannot drag and drop items, add redlining, add properties, use command buttons, and so forth, to the generated drawing, even though the commands are still enabled.